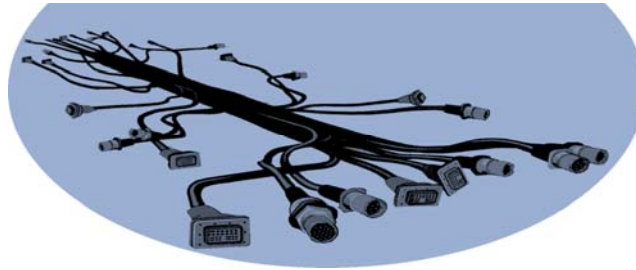


HarnesSYS™



Volume 5 - Integration

HarnesSYS™ - NX Interface

User Manual

HarnesSYS Version r20a
Document Version E



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Document Revisions History

Revision	date	Change Summary
E	December 2012	• Minor changes were made to correct errors. • Some chapters and topics were rearranged. • Added "jump links" between topics. • Reference: Mantis No. 7760.

Preface

Conventions

Italic text - is used to indicate a word or phrase, which has a special meaning with respect to **HarnesSYS™**, such as the name of a menu or option. Italic text is also used for examples. Smaller italic text is used for notes and helpful hints; these are also indicated by icons (see below)

Bold text - Bold text is used for emphasis a word, sentence or paragraph.

Bold italics - Bold italics is used for Procedure Heading.

COURIER - Text printed in Courier font represents text that appears on your screen.



Indicates note



Indicates an example.



Indicates a helpful hint.



Indicates additional information.



Indicates cautionary text or a warning. Both types of text are framed in a box. A Warning, however, is printed on a raster (gray) and has the title "WARNING!".



Indicates an operation that you, the user, are to perform. Operations in a series are also numbered.



Indicates a key on your keyboard.



Indicates a path to take. This usually entails selecting a number of options from menus.



Indicates a time consuming operation.



Indicate Procedure start



Indicates Procedure continue



Indicates what you should see on the screen - the computer's response.



Link to other document.



Indicates the end of the user guide.

About this User Manual

This User Manual describes the **HarnesSYS™** - NX Interface Process which enables the Design and Maintenance of Integrated Electrical-Mechanical Bundle Models.

Audience

This manual is for experienced designers. It explains the Interface Process through all its phases. However it is not intended to guide the user how to design and manufacture his Project.

Related Documents

This manual is one of the set of manuals that make up the **HarnesSYS™** user documentation.

HarnesSYS™ user documentation is organized in the following volumes:

- Volume 0 - Getting Started
- Volume 1 - Schematic Wiring Diagram (SWD)
- Volume 2 - 2D Routing/Jig
- Volume 3 - Wires
- Volume 4 - Parts
- Volume 5 - Integration
- Volume 6 - Project Management
- Volume 7 - Manufacturing.
- Volume 8 - Generating Technical Publications.

- Volume 9 - Retrofit.
- Volume 10 - Utilities
- Volume 11 - Importer
- Volume 12 - Miscellaneous

Document Organization

This manual includes the following chapters:

Chapter Number	Title	Description
Chapter 1	Introduction	
Chapter 2	Installation of HarnesSYS™ – NX Interface	This Chapter explains how to install the USER FUNCTION.
Chapter 3	NX Implementation of Bundle Structure	Diagram describing the bundle structure.
Chapter 4	HarnesSYS™ - NX Interface Process Flow	Describes the data flow for concurrent engineering using HarnesSYS™ and NX.

1 Introduction

In This Chapter:

Aim and Sub-aims of the HarnesSYS - NX Interface Process.	1
Terminology	2

HarnesSYS™ bundles may have 3D geometric representations in external mechanical CAD systems such as NX. This information exists in the mechanical CAD model as part of a digital Mock-up. These bundle models may be created in NX with the **Routing Electrical Application**.

The **HarnesSYS™** - NX Interface enables concurrent engineering by transferring data in both directions between **HarnesSYS™** and NX. **HarnesSYS™** requires geometrical data such as wire lengths and components locations. NX requires Parts data and segments diameters.

The interface is activated via an icon that activates the **HarnesSYS™** - NX Interface USER FUNCTION in NX.

1.1 Aim and Sub-aims of the HarnesSYS - NX Interface Process.

Aim

- Design and Maintain Integrated Electrical-mechanical Bundle Models in a concurrent engineering environment.

Sub-aims

- Provide a comprehensive integration methodology and process
- Combine the strength of both systems: Logical Data in **HarnesSYS™**, geometric data in NX.
- Fulfill mechanical and electrical requirements.

1.2 Terminology

The following are unique terminology for this user manual:

Basic Term	Description
BTP (Build-To-Package)	HarnesSYS™ document that contains all the engineering data for bundle manufacture using sub-documents such as Wire List, Part List, 2D-Routing and JIG .
Bundle	Wires and parts that are grouped together to be assembled as one unit using a JIG .
NX-Interface Export folder	The folder where the Interface creates files that transfer geometrical data from NX to HarnesSYS™ . It should be the same folder as HarnesSYS™ Import folder and should be defined by environmental variables prior to NX activation.(see the Installation chapter)
NX-Interface Import folder	This folder contains files that transfer Parts List and segment diameter data from HarnesSYS™ to NX. It should be the same folder as HarnesSYS™ Export Folder and should be defined by environmental variables prior to NX activation. (see detail) (on page 5)
Centerline Object	A basic routing object such as a single line, spline, Routing Segment, etc.
Clamp Marker	A point with text that is marked on the manufactured bundle at a specified location.
Connector Keyway	A line in a circular connector that represents the keyway direction from the point indicating the wire path termination . This line is used to calculate the clocking angle symbol for angled backshells on the JIG.
Device	Auxiliary item such as a clamp, backshell or protective device that is produced with the bundle.
End-Connector	Wire-End terminator that has a unique name on the aircraft (Ref. Des.).
Environmental variables	Variables that are defined in the Window Operating System for the Interface Program installation. These variables may be defined while a central NX script is activated or independently on each workstation.
HarnesSYS™ Export folder	This folder contains files that are used to transfer Parts data and segment diameters from HarnesSYS™ to NX. It is defined in HarnesSYS™ in the Project Defaults Form. It should be the same as the NX-Interface Import folder .
HarnesSYS™ Import folder	This folder contains files that are used to load NX geometrical data into HarnesSYS™ . These files are created by the Interface . It is defined in HarnesSYS™ in the administrator project default form. It should be the same as the NX-Interface Export folder .

Basic Term	Description
names for NX Objects	These naming conventions are set on NX lines and points to define the wire paths and other attributes inside standard Electro-Mechanical parts.
JIGHarnesSYS	A HarnesSYS™ Form Board drawing document that is used for bundle manufacture.
Node	Bundle segment junction point.
Note	A marker item that is produced within the bundle and is represented in the NX model by a point.
Parts List	A list of primary parts and their backshells belonging to a specific bundle in HarnesSYS™ .
Protective Device	A covering that is applied to the bundle such as a metal over-braid, sleeve or boot. It is represented in NX by points having the same NX parameter. These points indicate the device edges on the bundle segments.
Segment	A section of the bundle with uniform wire content.
Widely Spaced Terminator	A Widely Spaced Terminator is a single reference designator comprising several independent wire paths representing HarnesSYS™ terminator pins. Within NX, This is group of NX parts or Designator Points representing pins of the same End-Connector , where each pin has its own wires path.
Wire Path Routing	A 3D model representation of the bundle routing between its End-Connectors
PLMXML file	A file in PLMXML format that is used to transfer Part data from HarnesSYS™ to NX...
3d_asc	A file with the 3d_asc extension which is created by the Interface , and contains geometrical data of the NX bundle. This file is used by HarnesSYS™ to build the JIG or transfer the diameters back to NX.

2 Installation of HarnesSYS - NX Interface

In This Chapter:

Copy Files	5
Set environmental variables	6
Install HarnesSYS Interface Icon	8
Check the Installation of HarnesSYS - NX Interface	13
Check HarnesSYS Import & Export Folder Definitions	15

This installation is suitable for NX6 or NX8 with and without TeamCenter 8.

2.1 Copy Files

The Interface Program Package for NX contains the following files. After the **HarnesSYS™** installation process, these files should appear in the **~harness/r20a.00/NX interface** folder.

- NXinterface/hr_nx6_win7_64_interface.dll
- NXinterface/hr_nx6_win7_32_interface.dll
- NXinterface/hr_nx6_win7_TC8 32_interface.dll
- NXinterface/hr_nx6_win7_TC8 64_interface.dll
- NXinterface/hr_nx8_win7_64_interface.dll
- hr ugnx6 TC2007 interface.dll (this module works with Windows XP and TeamCenter 2007)
- hr ugnx6 interface.dll (this module works with Windows XP without TeamCenter)

This is the NXOPEN module (User Function) for Windows XP, 32 or 64-bit Windows7 which is executed by the user from the **HarnesSYS™** icon in NX.

The user should select the suitable file to his working environment.

2.2 Set environmental variables

Environmental variables define file locations and parameters for the **Interface Program**.

A. For computer sites with central system support for NX

When **NXOPEN module** is activated, the environmental variables will be already defined by the NX system Group. This setting requires one installation per site only.

B. For a single user definition or computer sites where NX management is a performed outside the Computer Center.

Click on the "Start" Microsoft Window button and choose the following menus:

→→ **Start** → **Setting** → **Control Panel** → **System** → **Advanced** → **Environmental Variables**

Before you start NX, install the following compulsory environmental variables:

Table 1: Environmental Variables

Environmental Variable Name	Environmental Variable Description
HR_UG_ROUTE_MASTER_STOCK_PRT	Path of *.ptb file that contains the PART_NAME definition of the *.prt file loaded by NX, in order to define stocks in NX Routing Electrical. The *.prt file must be installed or located previously. The *.ptb file location: {UGII_EPLIB_TABLE_PATH}\wires_mm.ptb is defined by NX environmental variable in {UGII_ENV_FILE} .
HR_UG_ROUTE_APPL_VIEW	Define the path of the appropriate ugroute_elec_XX.xml, where XX is inch/metric. The value of HR_UG_ROUTE_APPL_VIEW should be equal to the value of {UGII_ROUTE_ELEC_APP_VIEW} which is defined by NX environmental variable in {UGII_ENV_FILE}.
HR_ROUTE_ELEC_INTERFACE_DIMENSION=INCH or HR_ROUTE_ELEC_INTERFACE_DIMENSION=MM	Defines what unit will be used in Variable: HR_UG_ROUTE_MASTER_STOCK_PRT

Installation of HarnessSYS - NX Interface

Environmental Variable Name	Environmental Variable Description
HR_UG_INTERFACE_EXPORT_DATA	Defines the export folder where files are stored by the Interface program. Example: C:\nx2harness\
HR_UG_INTERFACE_IMPORT_DATA	Defines the import folder where files read by the Interface program are located. Example: C:\harness2nx\

2.3 Install HarnesSYS Interface Icon

In order to install the HarnesSYS™ Interface icon  in the main NX toolbox follow the following steps:



Copy the file "**User.mtx**" from your NX customized environment to

C:\Document and Setting\<user>\Local Settings\Application Data\Unigraphics Solutions\NX\user.mtx



Open local (without customization) NX (from {UGII_ROOT_DIR}\ugraf.exe)



From the top menu Choose **Tools** → **Customize**



Choose **Commands** tab → Choose **New Button** → Choose **New User Command**.

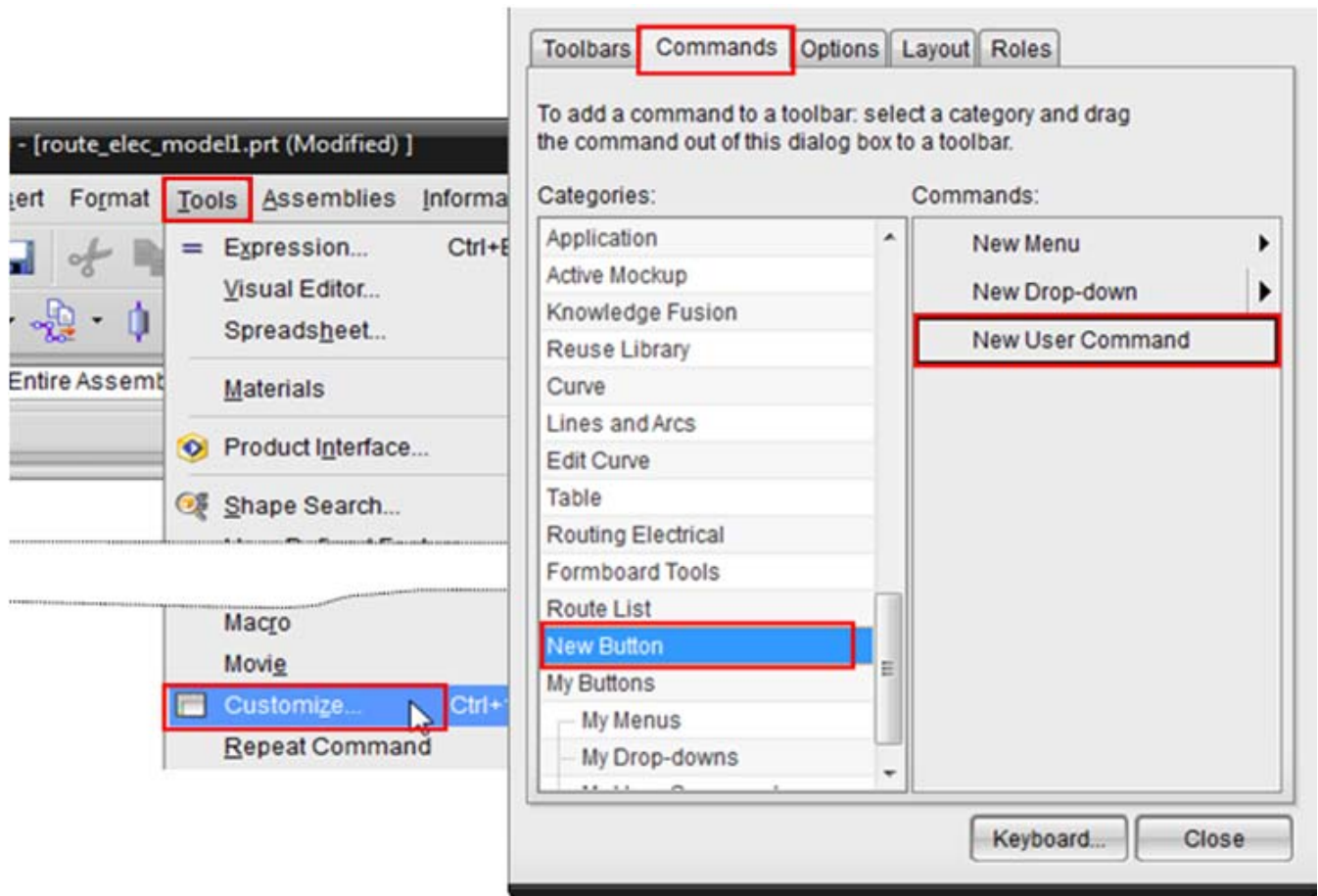


Figure 1: Customize Window



Do not close the **Customize** window for the next steps.



Create a command button by dragging **New User Command** to any location on the menu bar or toolbars.



Right click on the created button **User Command** and the following drop down menu appears in the displayed menu:

- Enter Name: HarnessSYS - The HarnessSYS string will appear in the menu bar instead of User Command.
- Press Enter to save the new command name.
- Again, right click on the User Command button and the following drop down menu appears:

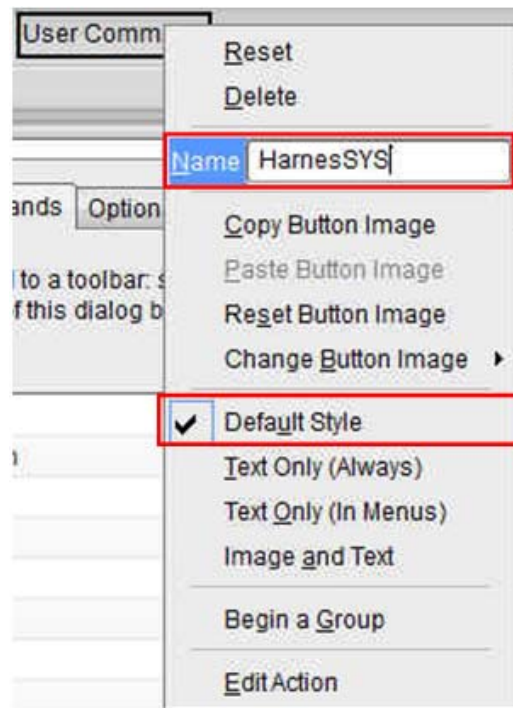


Figure 2: User Command window

→ Choose: **Default Syle**



Select **Change Button Image** option → Choose Image Name option.

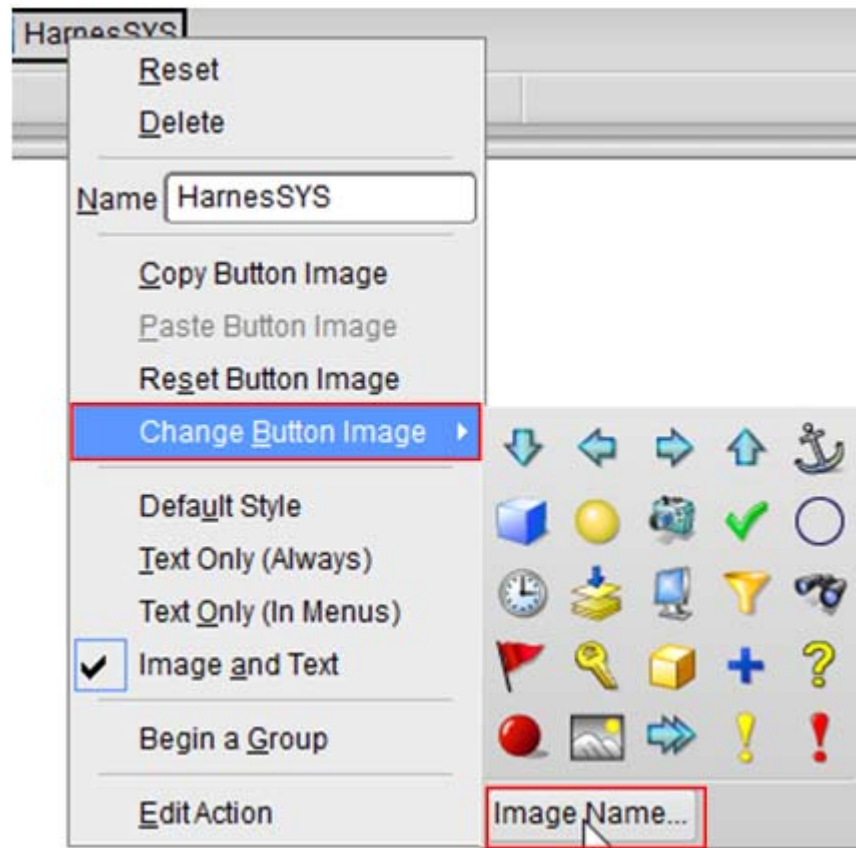


Figure 3: Change Button image window



Browse to choose IAI_HarnessSYS.bmp (The IAI_HarnessSYS.bmp is part of the Manual Installation).

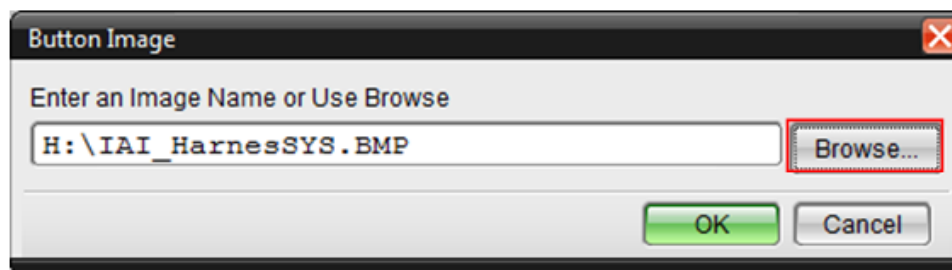


Figure 4: Button image window



Press OK to confirm.



As a result, the **HarnessSYS™** icon  appears in the Command Properties area.



Click the **Edit Action** button option on the **HarnesSYS™ icon** in the Commands area.

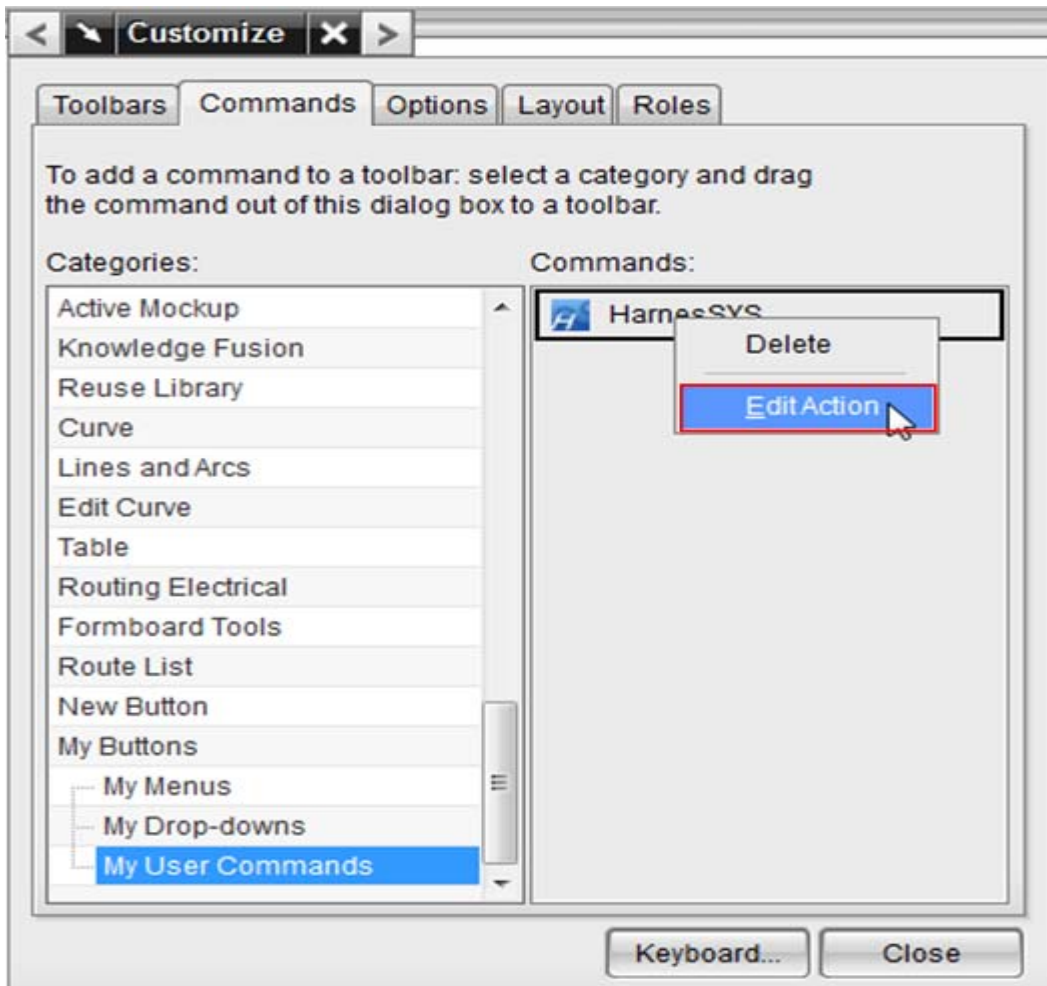


Figure 5: Customize window



The following menu will appear.



Figure 6: Button action window



In the "Enter an Action or Use Browse" text box:

- Browse for the following .dll
~harness/rxx.xx/NXinterface/hr_nx8_win7_64_interface.dll
- In the "Enter Button Message text" box enter: **HARNESYS**
- Press OK to confirm.



After you finish defining your function

- Close NX
- Copy C:\Documents and Settings\<user>\Local Settings\Application Data\Unigraphics Solutions\NX\user.mtx to its place in your customized environment.



Enter NX with your customized environment and check your new function.



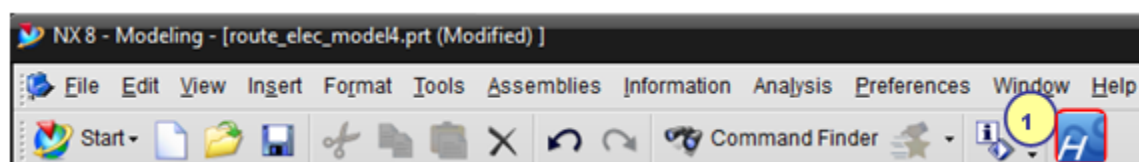
The Process above enables the user to run the **HarnesSYS™** Interface.

2.4 Check the Installation of HarnessSYS - NX Interface

This check shows the Creation Date of the Interface and the environmental variables status as they appear after the **HarnesSYS™ - NX Interface installation** (on page 5) .

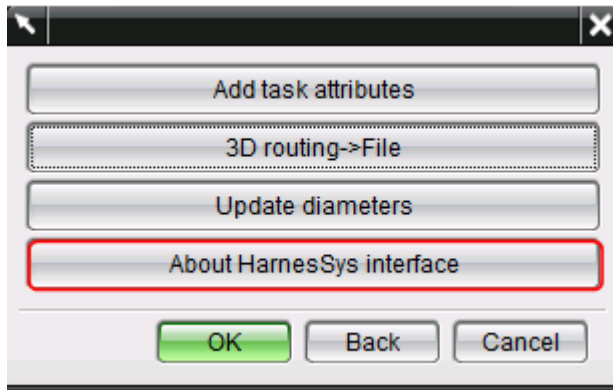


Activate the **HarnesSYS™ Interface** using the **H icon** on the NX work space.





The **THarnesSYS™ – NX Interface** main window appears:

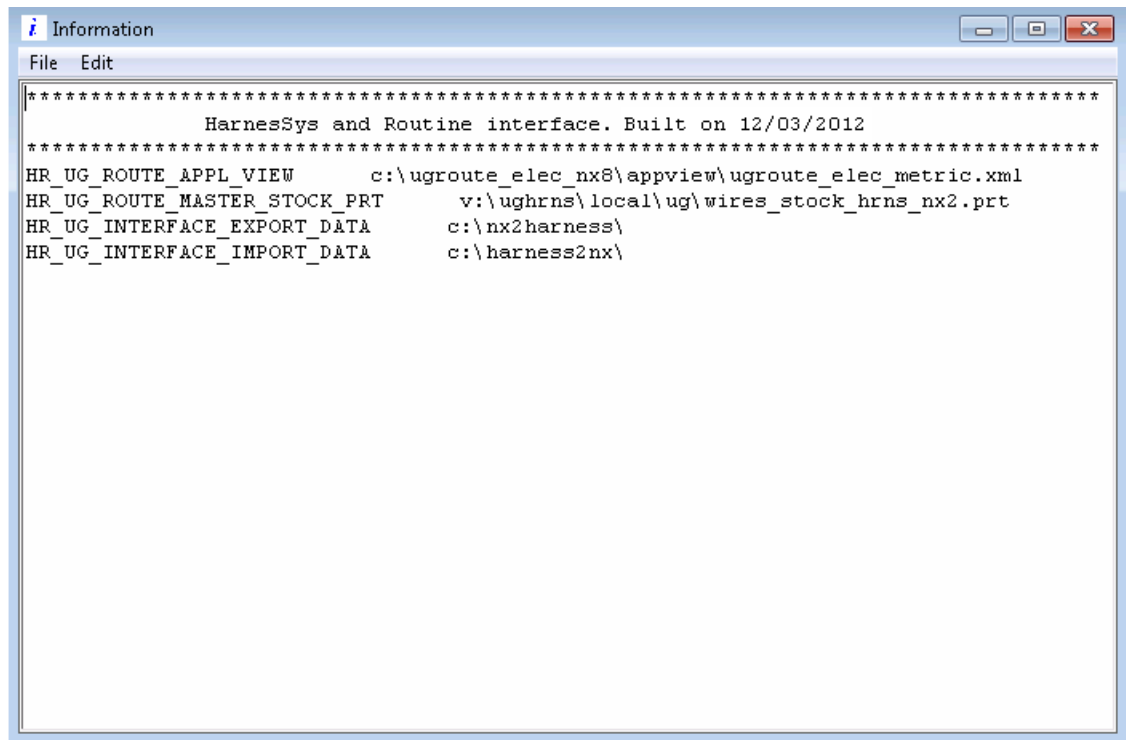


Choose the **About THarnesSYS™ Interface** option.

This option provides creation date of the module and all possible user-defined environmental variables of the **THarnesSYS™** Interface with their current values. If a variable has not been defined in the current session it will be shown with an empty value.



The following menu appears:



2.5 Check HarnesSYS Import & Export Folder Definitions



Log in to **HarnesSYS™** account.



Run **HarnesSYS™**.



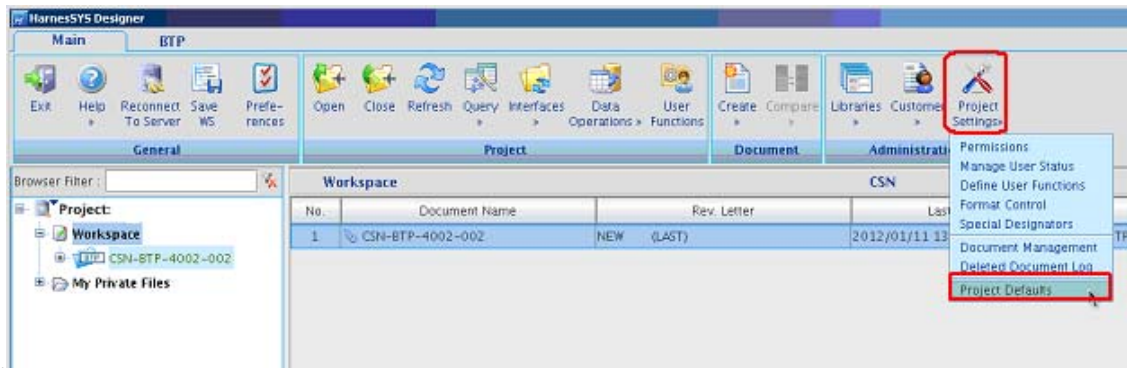
→ **Administration** → **Project Settings**



The **HarnesSYS™** start Session Window appears (with the New Project button enabled).



Click → **Project default**



The "Set Project Default for Project: *Project Name*" window appears:

Set Project Default for Project:

Administration Graphics Wires Parts Reports Manufacturing Tech Pub Release / PreRelease Plot Retrofit FFD

Lower-case Pin Names

Sign Location: Prefix
Letter Sign: (*) Star

General

Cage Code:
Length and Weight System: FPS
Pin Reservation Limit: 9999 days
Round Length To Nearest: 0.0 inch
Separation Code: From Wire Name
Multi Path Allowed: ☐
Auto Open BTP and Link Current Components: ☒
Use Default Effectivity: ☐ Effectivity...
Using Cascade Base Numbers Convention: ☐
Update Weight By: Spec. Libraries/Wt. Assign.
Mandatory Authority Assignment Prior to Document Pre-rel/Release: ☐

Document

Point Release Enabling: Not Allowed
Skipped Letters (Rev/Branches): IOQ
Initial Rev. Letters of New Doc.: NEW

File Path

Import XML/3D ASCII: /nx2harness/
Export XML/3D ASCII: /harnes2nx/
Export BTP-to-PDM: /tmp/ BTP file list

Activate Work

Sequence: ☐
Stage: ☐

Load Analysis

Interval (Sec)	AC	DC
I	5	5
II	300	300
III	900	900

OK Cancel

Figure 7: Project Default window



Set the "**Export XML/3D ASCII**" and "**Import XML/3D ASCII**" paths to the same (shared) folders as the *NX Interface Import and Export folders (on page 6)* To change **HarnesSYS™** Export and Import Folders Definition, The User should have **HarnesSYS™** Project Administrator privileges.

b Release / PreRelease Plot Retrofit FFD

Document

Point Release Enabling Not Allowed

Skipped Letters (Rev,Branches) IOQ

Initial Rev. Letters of New Doc. NEW

File Path

Import XML/3D ASCII /nx2harness/

Export XML/3D ASCII /harnes2nx/

Export BTP-to-PDM /tmp/ BTP file list

Activate Work Load Analysis



Press **OK** to confirm the changes.

3 NX Implementation of Bundle Structure

The following NX model and its components should be built in order to transfer data between NX and **HarnesSYS™** using the **Interface**.

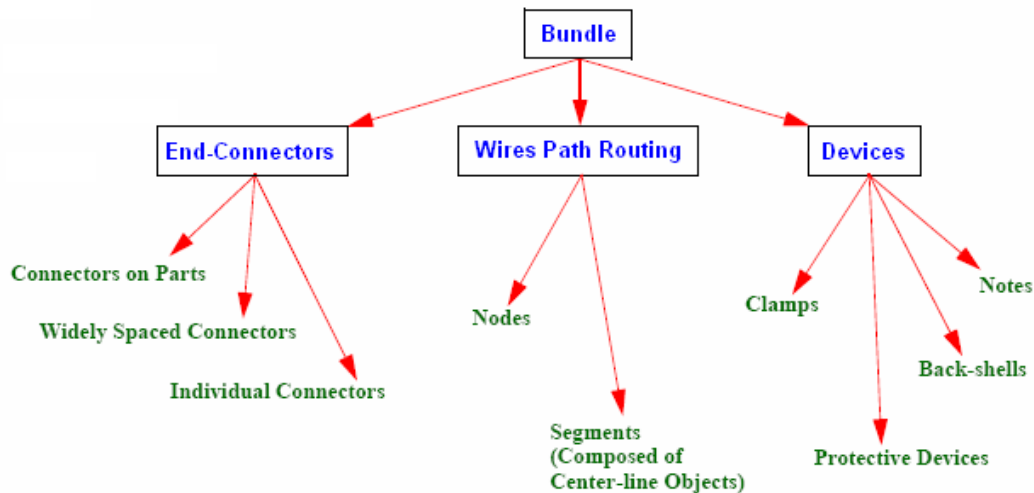


Figure 8: Diagram describing the bundle structure

Each **bundle** is placed in a separate assembly component. The bundle contains parts that are manufactured with the bundle as well as the bundle geometry. In addition the bundle may optionally contain sub-assemblies (e.g. an isolated routing zone in the aircraft).

Parts that are manufactured with the bundle are typically **end-connectors** such as circular connectors, rectangular connectors and accessories such as lugs, ground modules, rail modules, round modules, switches, Coax/Triax splices and Coax/Triax connectors. Circular connectors may contain **Connector Keyway** locations represented by line.

These parts may be also **devices** such as **backshells** and **adapters**, External parts such as clamps are not **included in the NX assembly** but they may have trace markers on the Jig through NX points.

Prepare Electrical and Geometrical Data for NX Part (on page 26) Models shows the NX implementation process.

The bundle geometry is represented by NX Routing Segments, lines and points. When several segments are connected to the same point, each of them should contain different NX centerline objects. This data structure may be created by the NX Routing Segments Electrical Application. In addition the bundle geometry may contain NX points that represent **Designator Points**, **Widely Spaced Terminators**, **notes** or **protective devices**. These points are to be installed on the Jig for the bundle manufacturing.

** The User should be familiar with NX Electrical terminology and activities.*

4 HarnesSYS - NX Interface Process

The products of this process:

1. BTP in ***HarnesSYS™***.
2. Updated model in NX.

The following scheme describes the data flow for concurrent engineering using ***HarnesSYS™*** and NX.

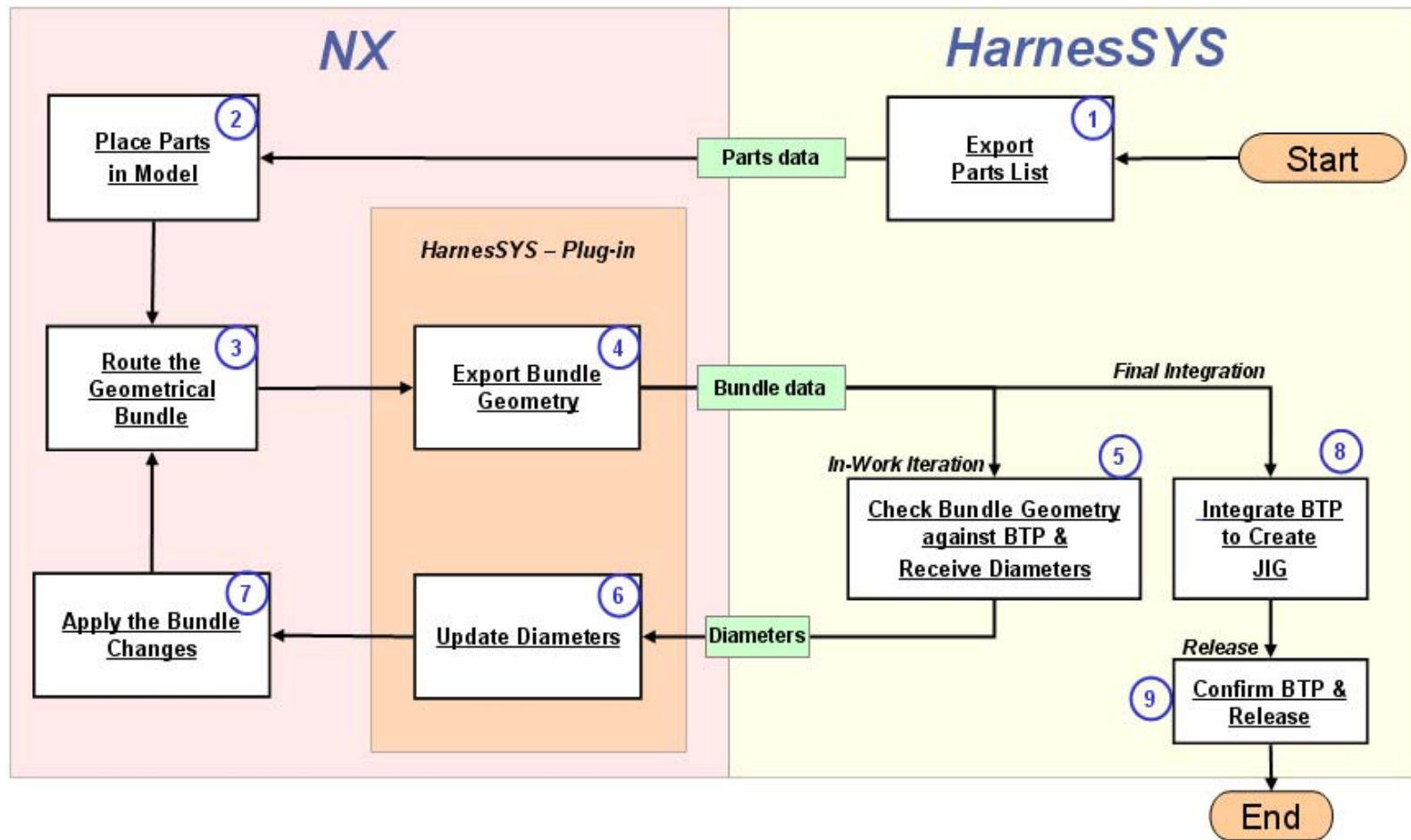


Figure 9: Data flow

 List of Links:

1. Export Parts List.....	21
2. Place HarnesSYS Parts in Model	25
3. Check Bundle Geometry against BTP & Receive Diameters.....	87
4. Update HarnesSYS Diameters	94
5. Apply the Bundle Changes.....	96
6. Integrate BTP to Create Jig.....	98
7. Confirm BTP and Release	99

4.1 Export Parts List

 List of Links

Check HarnesSYS Import & Export Folder Definitions	15
Check the Installation of HarnesSYS - NX Interface	13
Export HarnesSYS Parts.....	21

4.1.1 Export HarnesSYS Parts

This function creates a PLMXML file containing **Designators and lugs**.

Designators are extracted from Part occurrences. This list contains only major parts and backshells.

Lugs are extracted according to term codes and common terminators that may appear on the wires at both ends.



The export process relates to term codes that are stored directly on the wires.

Term codes may be assigned to wires automatically using "Assign Terminator codes" BTP option or manually using "Wire Single change" or "Group Change" on the Wire List.

Please note that you may see term codes on the wire reports that are calculated on the fly, but they will be not included in the PLMXML file generated during the export process.

Example: Process to export Parts & Lugs



In *HarnesSYS*™, choose **Interfaces** → **Export** → **NX PLMXML - Parts & Lugs**

or click (MB3) on the selected **BTP** from the Workspace.



Choose the "**Export** → **NX PLMXML - Parts & Lugs** " option

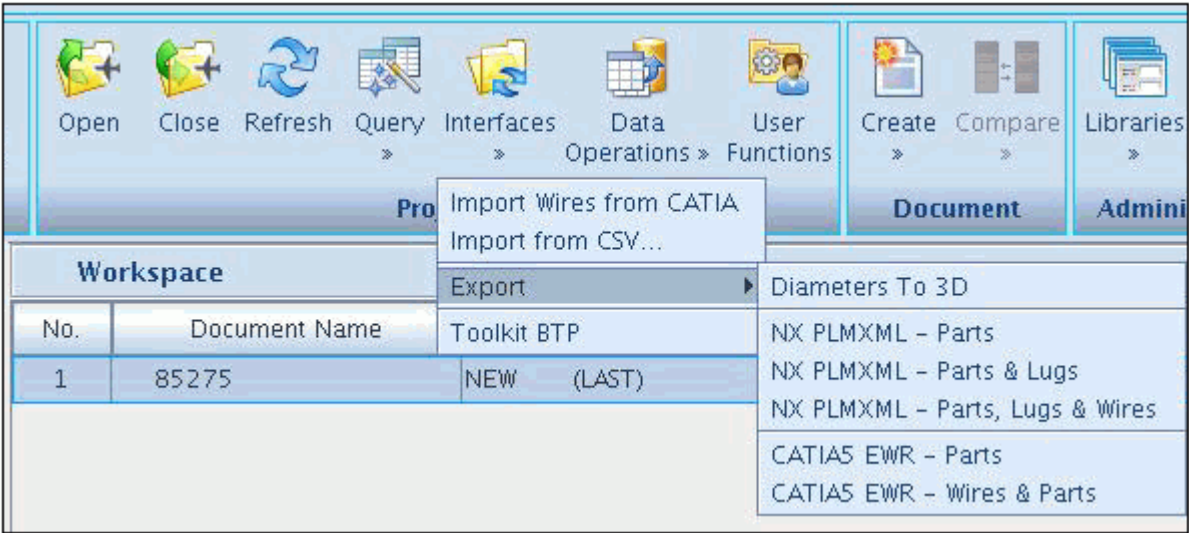


Figure 10: Export HarnesSYS™ Screen



If there are no errors during the process, the following termination message will appear:

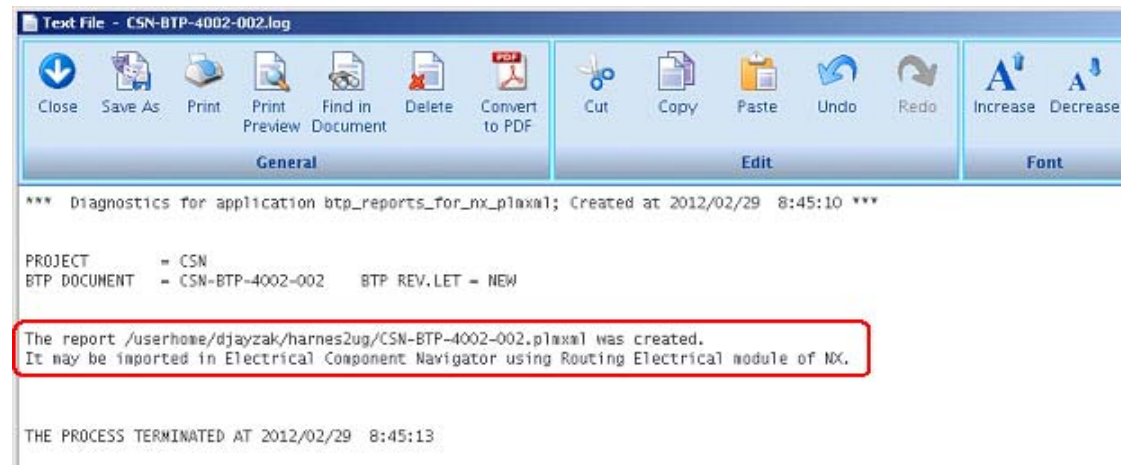


Figure 11: Termination message window

If the process terminated successfully, the file <BTP-Name> .plmxml* will be created in the **HarnesSYS™ Export folder**.

If "Export → NX PLMXML - Parts" option is used, the <BTP-Name> -PL. plmxml* file will be created in the **HarnesSYS™ Export folder**.

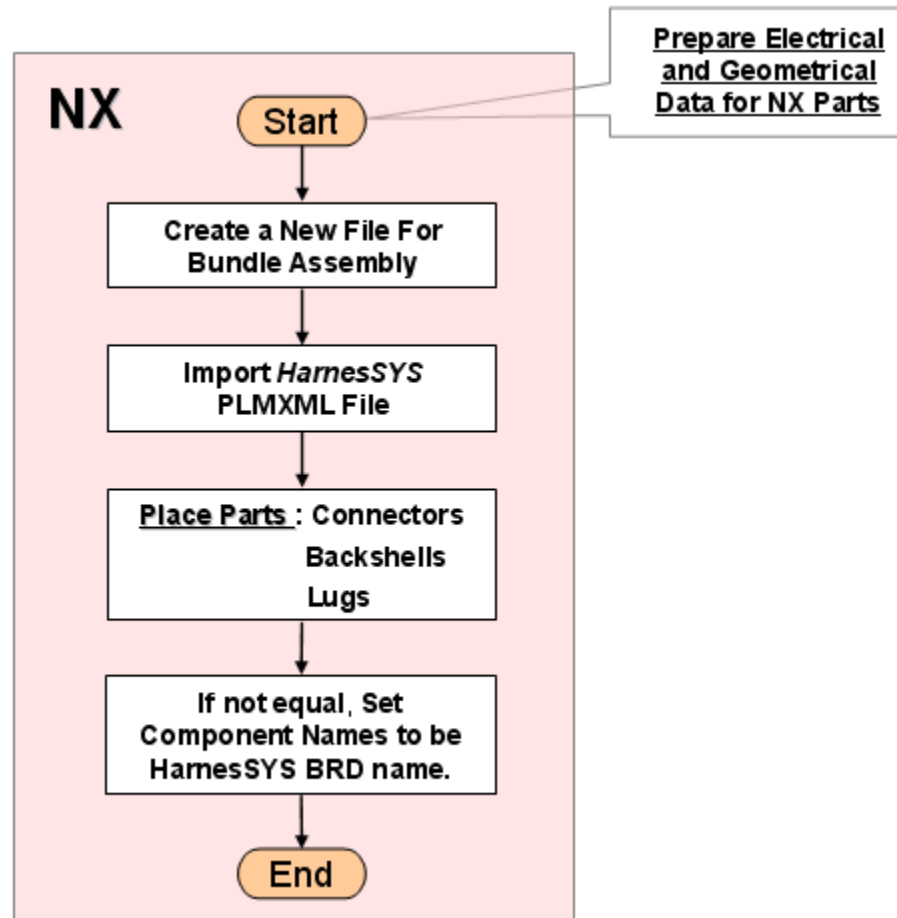


If the **HarnesSYS™ Export folder** and the **NX-Interface Import folders** are not shared folders (i.e. the file created in **HarnesSYS™ Export folder** does not automatically appear in the **NX-Interface Import folder**) then you must manually copy the files created by **HarnesSYS™** in the **HarnesSYS™ Export folder** to the **NX-Interface Import folder**.

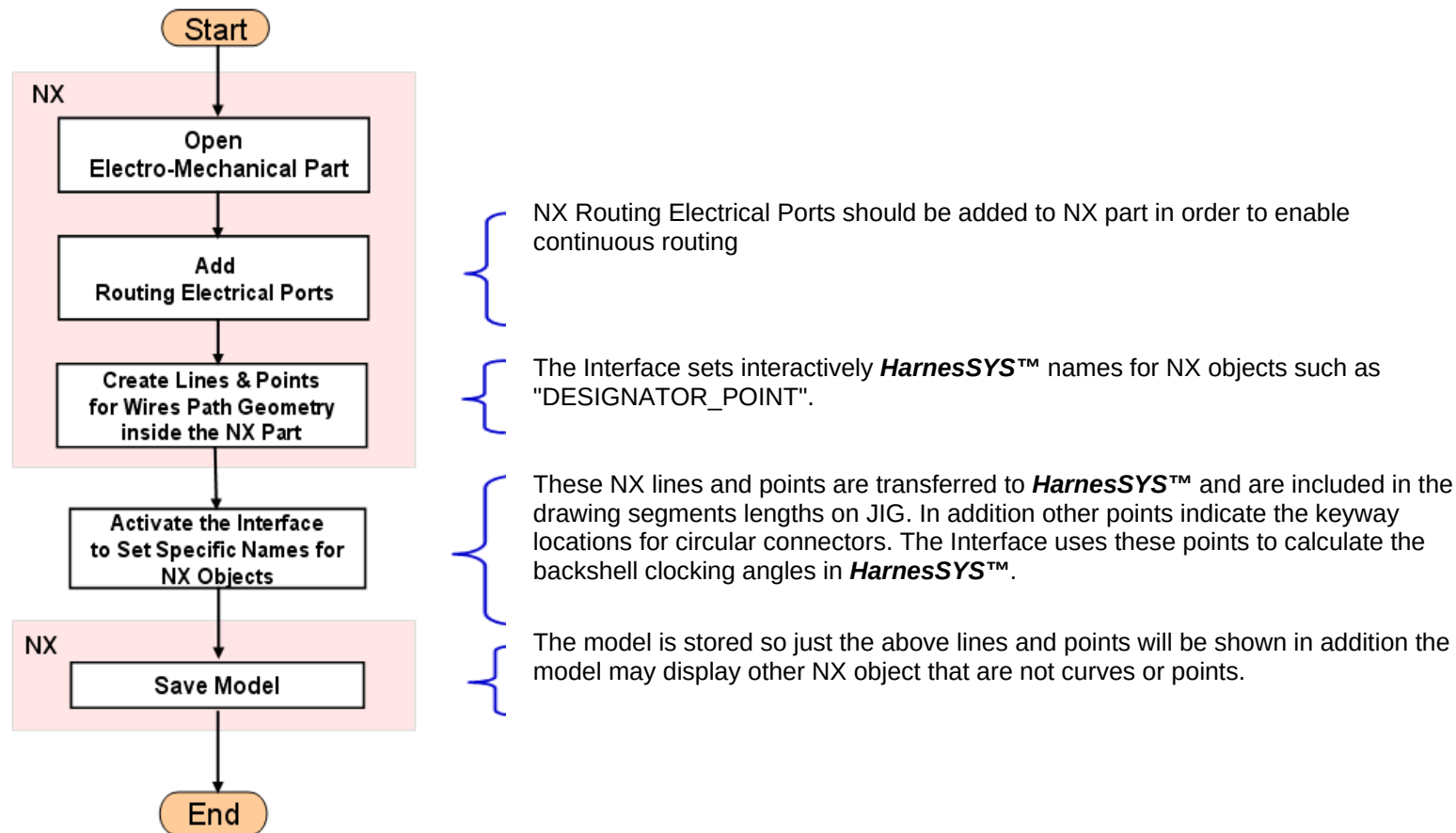
* PLMXML - An open XML-based file format

4.2 Place HarnesSYS Parts in Model

The bundle assembly contains predefined electro-mechanical NX parts. These parts contains specific feature for installation and wire path definition. See instructions how to Prepare Electrical and Geometrical Data for NX Parts. After the parts are defined, follow NX documentation on Assemblies or Electrical Routing applications according to the following work flow:



4.2.1 Prepare Electrical and Geometrical Data for NX Parts



The help lines are invisible. This is the default displayed Reference set.

4.2.1.1 Basics

The **HarnesSYS™** - NX interface declares and expects to find some lines, points, ports geometry and part attributes for each Electro-Mechanical part. These lines represent wires path or other features (e.g. keyway).

Most lines represent a portion of the wire path inside the various parts. The proper positioning of their end points is critical for the correct behavior of the interface. In each part, these line segments shall form a continuous path.

When several parts are connected together one another (e.g. "connector with backshell", "connector, adapter and backshell"), the continuity of these various lines shall be respected without any overlap or gap between the lines.

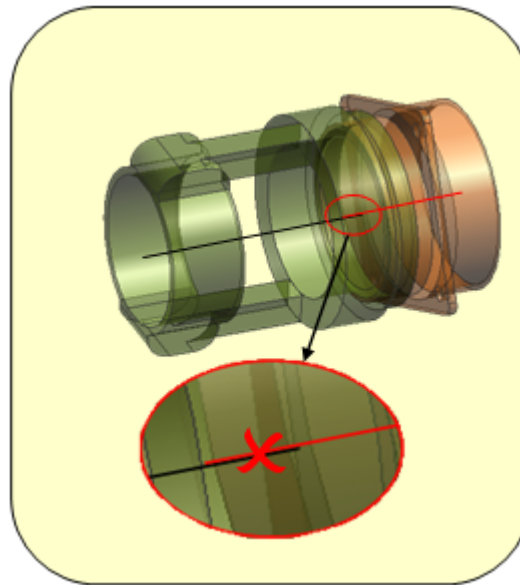


Figure 12: Check the continuity of special lines

The ports specified in this document are necessary for the **HarnesSYS™** - NX interface. The ports shall be located at the end points of the lines that define portions of the wire paths. It is assumed that the library parts will include additional ports for the placement of the parts in NX.

All lines and points should be visible (in the context of the cable assembly) in order to run **HarnesSYS™** - NX interface. The interface does not recognize hidden lines and points. Please make sure that all other lines and points in the NX model are hidden. Use the proper NX reference set to control the visualization of the required geometry.

HarnesSYS™ expects predefined special geometry for the following types of parts:

Connectors

- a. **Basic connector** (on page [30](#))
- b. **Circular connector** (on page [32](#))
- c. **Coupler** (on page [34](#))
- d. **Rectangular connector** (on page [36](#))
- e. **Widely-spaced terminator** (on page [38](#))

Devices

- a. **Circular angled backshell** (on page [42](#))
- b. **Rectangular angled backshell** (on page [44](#))
- c. **Straight backshell** (on page [40](#))
- d. **Adapter** (on page [46](#))

Embedded clamp (on page [48](#))

4.2.1.2 Connectors

 List of Links

Basic connector structure.....30

Circular Connector Structure.....32

Coupler part Structure.....34

Rectangular Connector Structure.....36

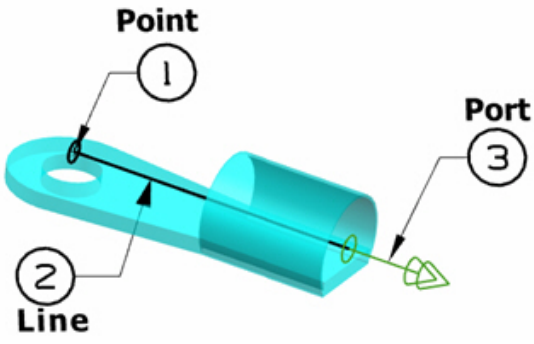
Widely spaced terminators on single part Structure.....38

Basic connector structure

Basic connectors are parts that are directly connected to incoming wire segments, such as lugs, relays, switches, mod-blocks (i.e. without backshell, without adapter...).

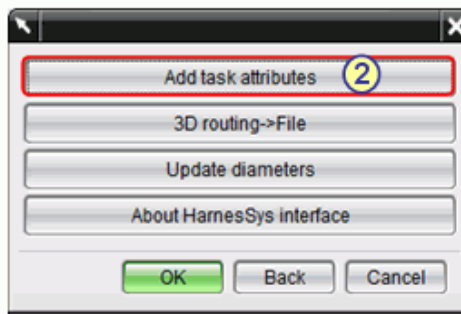
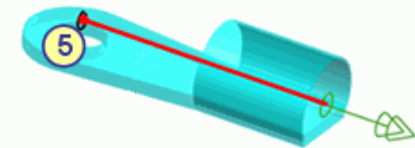
Each basic connector is defined by a NX Solid Body representing its geometry. In addition it should contain one point, one line and one multi port.

Table 2: Representation of Basic Connectors by NX objects

	①	The point indicates the wires termination point inside the connector.
	②	The line indicates the wires path inside the basic connector. It goes from the termination point to the multi port.
	③	The optional multi-port is located at the extremity of the line.

Add Task Attribute to Basic connector

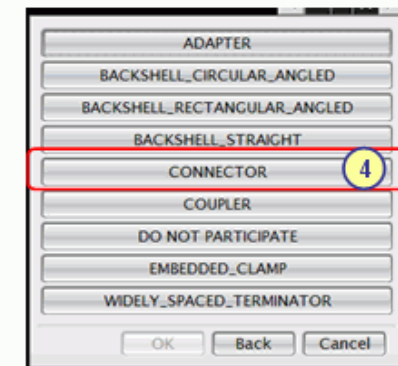
- ① Click on the  icon to open the "HarnesSYS Interface" window.
- ② Choose "Add task attributes", to open the **Add task attributes** window.
- ③ Press the "Part attributes" option to open the **Part attributes** window.
- ④ In the window that appears, choose the "**CONNECTOR**" attribute.
- ⑤ In the **Add task attributes window** choose the "**Designator Point**" option.
- In the Selection Window that appears, choose the point.



Main Interface window



Add task attributes window



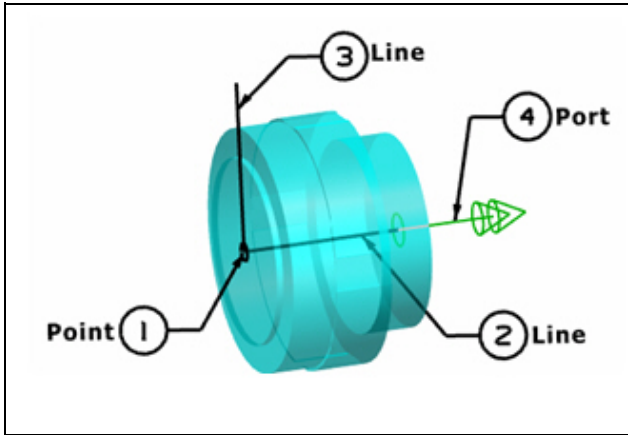
Part attributes window

Result: "HARNES_TASK" = "CONNECTOR" attribute is assigned to the NX part.
"HARNES_TASK" = "DESIGNATOR_POINT" attribute is assigned to the point.

Circular Connector Structure

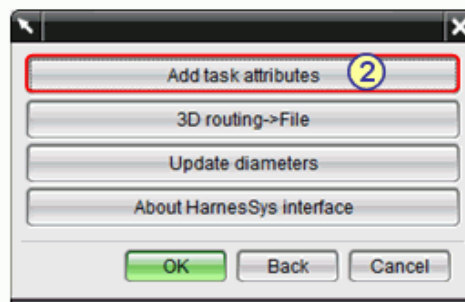
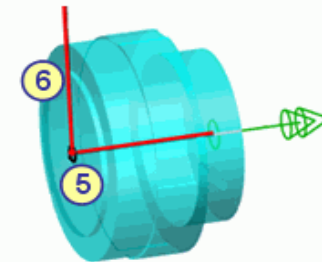
Each circular connector is defined by a NX Body representing its Shape. In addition it should contain one point and two lines.

Table 3: Presentation of Circular Connector by NX objects

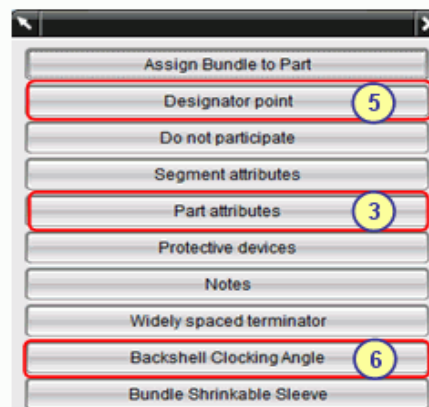
	1	The point indicates the wire termination point inside the connector.
	2	The first line indicates the wires path inside the connector from the wires termination point to the outer face of the connector.
	3	The second line indicates the keyway direction. It should be located on the same plane as the wires termination point and perpendicular to the connector axis.
	4	The optional multi-port is located at the extremity of Line 2.

Add Task Attribute to Circular Connector

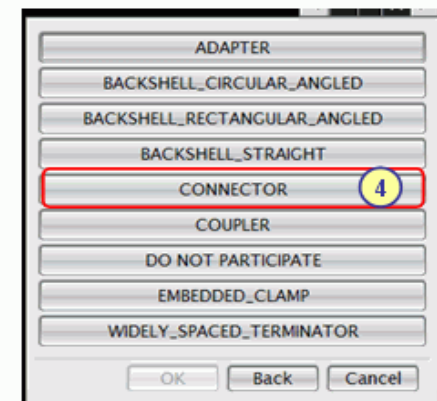
- ① Click on the  icon to open the "HarnesSYS Interface" window.
- ② Choose "Add task attributes", to open the **Add task attributes** window.
- ③ Press the "Part attributes" Option to open the **Part attributes** window.
- ④ In the window that appears, choose the "**CONNECTOR**" attribute.
- ⑤ In the **Add task attributes window** choose the "**Designator Point**" option.
- In Selection Window that appears, choose the point.
- ⑥ In the **Add task attributes window** choose the "**Backshell Clocking Angle**" option.
- In Selection Window that appears, choose the keyway line.



Main Interface window



Add task attributes window



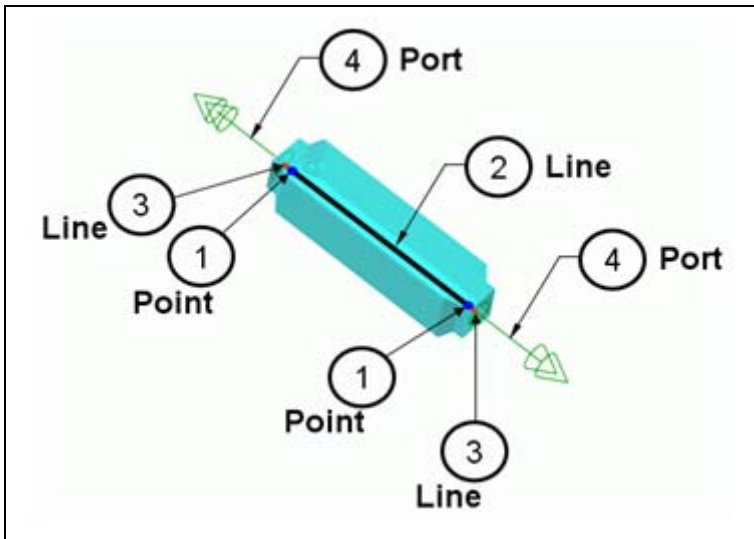
Part attributes window

Result: "HARNES_TASK" = "CONNECTOR" attribute is assigned to the NX part.
 "HARNES_TASK" = "DESIGNATOR_POINT" attribute is assigned to the point.
 "HARNES_TASK" = "BACKSHELL_CLOCKING_ANGLE" attribute is assigned to the line.

Coupler part Structure

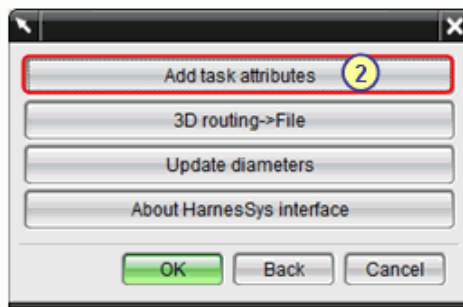
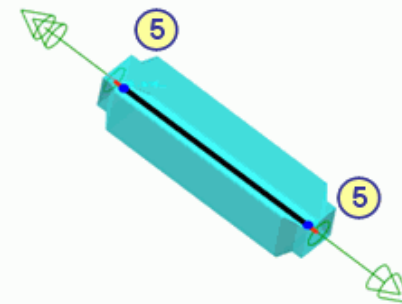
Each **Coupler** part is defined by a NX Body representing its shape. In addition it should contains three aligned lines and two end points of the middle line.

Table 4: Representation of Coupler by NX objects

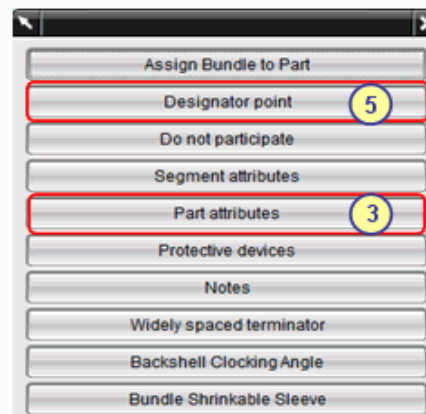
	1	Two points indicate the wire termination points inside the Coupler. Each point has a unique name (e.g. 1B...,2B...,3S)
	2	These lines represent the coupler on the jig (Form Board). No wires pass through these lines.
	3	The lines indicate wire path inside the Coupler, starting from a termination point.
	4	The optional multi-port is located at the extremity of each line.

Add Task Attribute to Data Bus Coupler

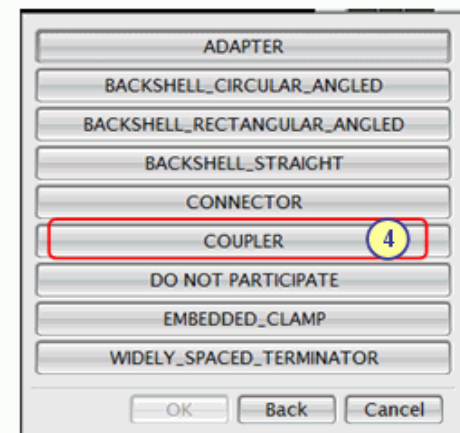
- ① Click on the  icon to open the "HarnesSYS Interface" window.
- ② Choose "Add task attributes", to open the **Add task attributes** window.
- ③ Press the "Part attributes" Option to open the **Part attributes** window.
- ④ In the window that appears, choose the "**COUPLER**" attribute.
- ⑤ In the **Add task attributes window** choose the "**Designator Point**" option.
- In Selection Window that appears, choose all of the wire path end points.



Main Interface window



Add task attributes window



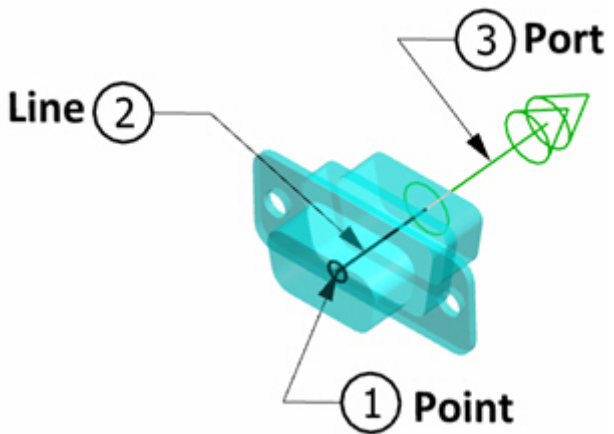
Part attributes window

Result: "HARNES_TASK" = " COUPLER" attribute is assigned to the NX part.
"HARNES_TASK" = "DESIGNATOR_POINT" attribute is assigned to the points.

Rectangular Connector Structure

Each **Rectangular Connector** is defined by a NX Body representing its shape. In addition it should contain one point, one line, one optional multi-port as follows:

Table 5: Representation of Rectangular Connector by NX objects

	①	The point indicates the wires termination point inside the connector..
	②	The line indicates the wires path inside the connector from the wires termination point to the outer face of the connector.
	③	The optional multi-port is located at the extremity of the line.

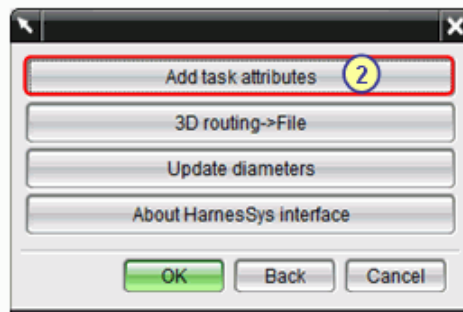
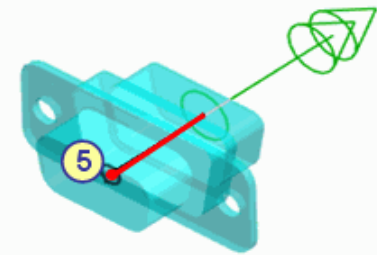


Special cases:

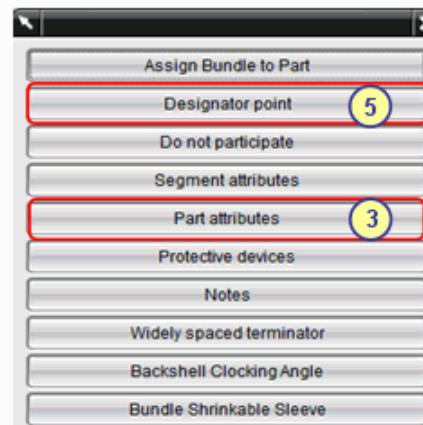
- One wire path and only one backshell if present.
- Several wire paths.

Add Task Attribute to Rectangular connector

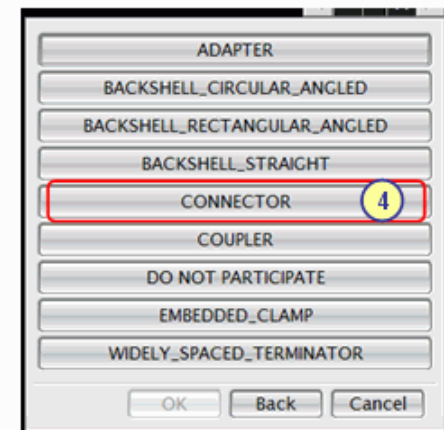
- ① Click on the  icon to open the "HarnesSYS Interface" window.
- ② Choose "Add task attributes", to open the **Add task attributes** window.
- ③ Press the "Part attributes" Option to open the **Part attributes** window.
- ④ In the window that appears, choose the "**CONNECTOR**" attribute.
- ⑤ In the **Add task attributes window** choose the "**Designator Point**" option.
- In Selection Window that appears, choose the point.



Main Interface window



Add task attributes window



Part attributes window

Result: "HARNES_TASK" = "CONNECTOR" attribute is assigned to the NX part.
"HARNES_TASK" = "DESIGNATOR_POINT" attribute is assigned to the point.

Widely spaced terminators on single part Structure

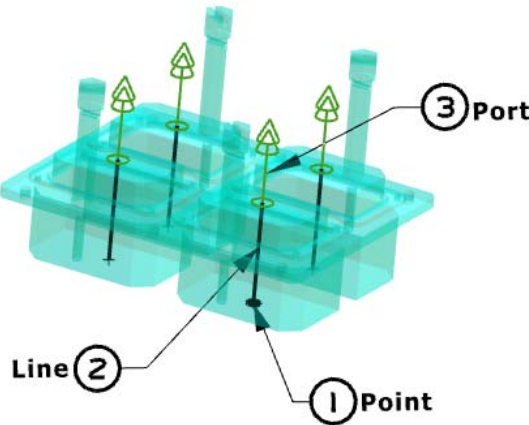
A widely spaced terminator part is a connector that has a single reference designator and contains several independent wire paths.

Each widely spaced terminator part is defined by a NX Body representing its shape. In addition it should contain one point and line per wire path; its definition is very similar to a series of rectangular connectors.

Each wire path is identified by a single character string. i.e. "A",

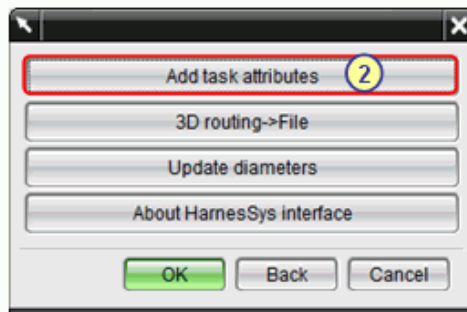
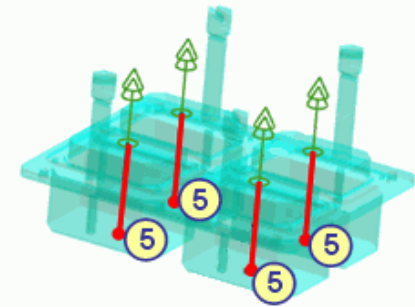
Each pin name reaching a given terminator is defined according to the following naming pattern: "XXX...", e.g. "A...", "B...", "1..." and so on.

Table 6: Representation of single part Widely Spaced Terminator by NX objects

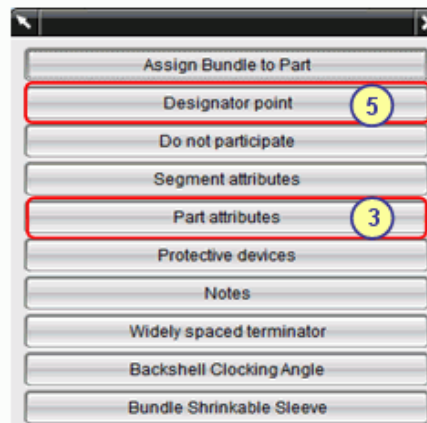
	1	For each wires path, a point indicates the wire termination point inside the terminator. Each point has a unique name (e.g. A...,B...)
	2	Each line indicates one wires path inside the Terminator, starting from a termination point.
	3	The optional multi-port is located at the extremity of each line.

Add Task Attribute to Widely spaced terminator on single part

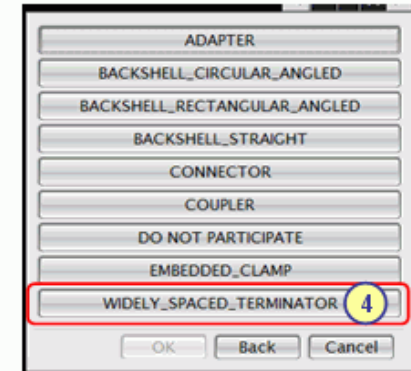
- ① Click on the  icon to open the "HarnesSYS Interface" window.
- ② Choose "Add task attributes", to open the **Add task attributes** window.
- ③ Press the "Part attributes" Option to open the **Part attributes** window.
- ④ In the window that appears, choose the "**Widely Spaced Terminator**" attribute.
- ⑤ In the **Add task attributes** window choose the "**Designator Point**" option.
- In Selection Window that appears, choose all of the wire path end points.



Main Interface window



Add task attributes window



Part attributes window

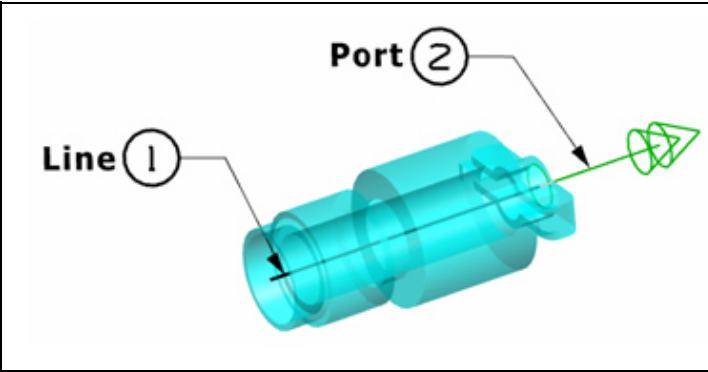
Result: "HARNES_TASK" = "Widely Spaced Terminator" attribute is assigned to the NX part.
"HARNES_TASK" = "DESIGNATOR_POINT" attribute is assigned to the points.

4.2.1.3 Backshells

Straight Backshell Structure

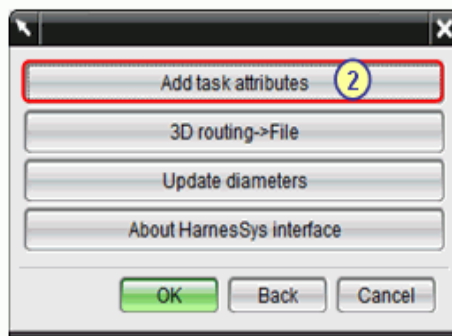
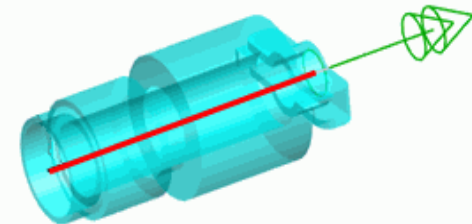
Each **Straight Backshell** is defined by a NX Solid Body representing its geometry. In addition it should contain one line.

Table 7: Representation of Straight Backshell by NX objects

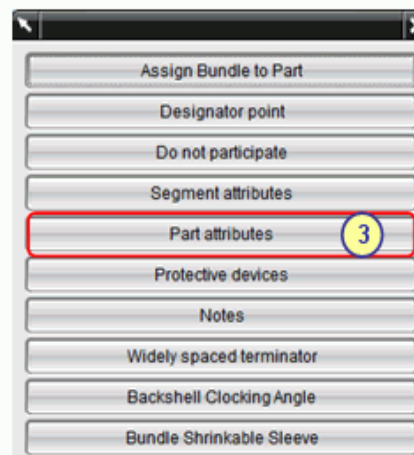
	1	The line describes the wires path in the backshell between the Connector part and multi port.
	2	The optional multi-port is located at the extremity of Line 1.

Add Task Attribute to Straight Backshell

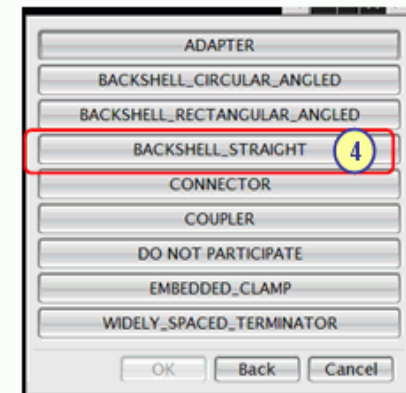
- 1 Click on the  icon to open the "HarnesSYS Interface" window.
- 2 Choose "Add task attributes", to open the **Add task attributes** window.
- 3 Press the "Part attributes" Option to open the **Part attributes** window.
- 4 In the window that appears, choose the **"BACKSHELL_STRAIGHT"** attribute.



Main Interface window



Add task attributes window



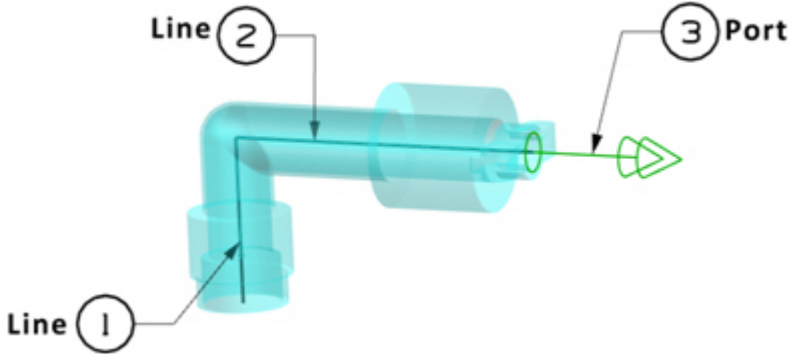
Part attributes window

Result: "HARNESS_TASK" = "BACKSHELL_STRAIGHT" attribute is assigned to the NX part.

Circular Angled Backshell Structure

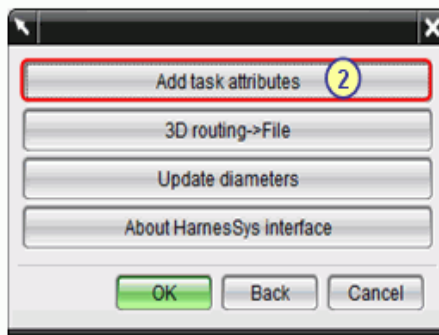
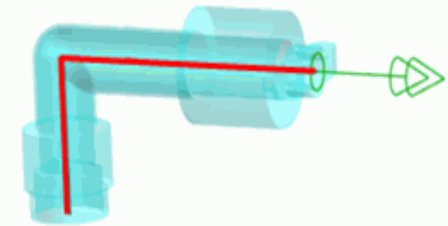
A **Circular angled backshell** is defined by a NX Solid Body representing its geometry. In addition it should contain two lines.

Table 8: Representation of Circular Angled Backshell by NX objects

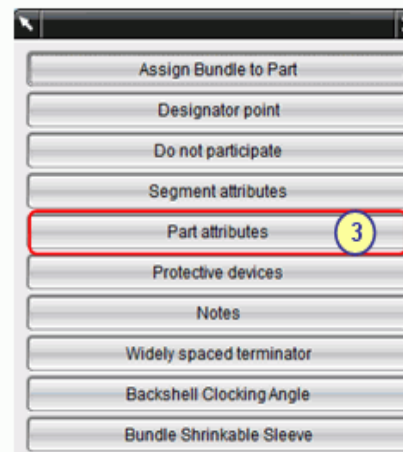
	1	These two lines together represent the wires path though the backshell. Line 1 is from the connector side to the angle of the wire path.
	2	Line 2 is from the angle to the outer face where wires are connected. These two lines share the same end point (at the angle).
	3	The optional multi-port is located at the extremity of Line 2.

Add Task Attribute to Circular Angled Backshell

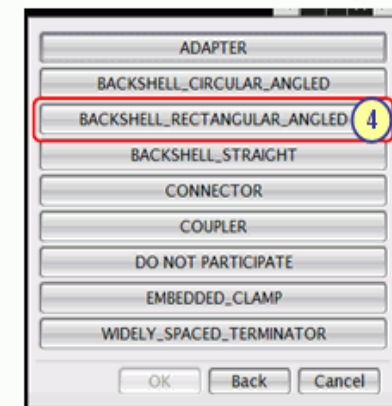
- 1 Click on the  icon to open the "HarnesSYS Interface" window.
- 2 Choose "Add task attributes", to open the **Add task attributes** window.
- 3 Press the "Part attributes" Option to open the **Part attributes** window.
- 4 In the window that appears, choose the "**BACKSHELL_CIRCULAR_ANGLED**" attribute.



Main Interface window



Add task attributes window



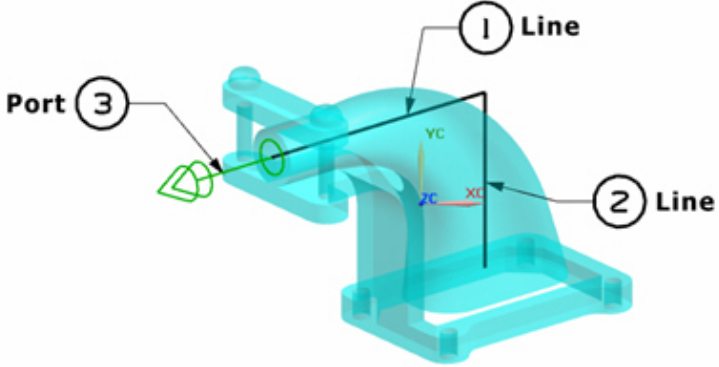
Part attributes window

Result: "HARNESS_TASK" = "BACKSHELL_CIRCULAR_ANGLED" attribute is assigned to the NX part.

Rectangular angled backshell Structure

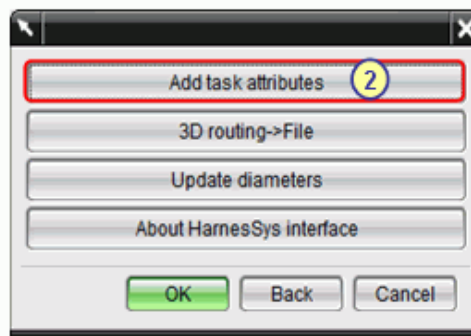
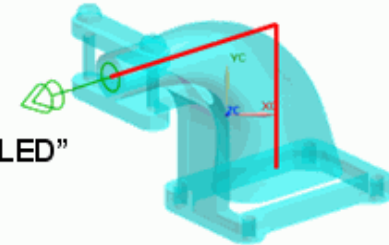
A **Rectangular Angled backshell** is defined by a NX Solid Body representing its geometry. In addition it should contain two lines.

Table 9: Representation of Rectangular Angled Backshell by NX objects

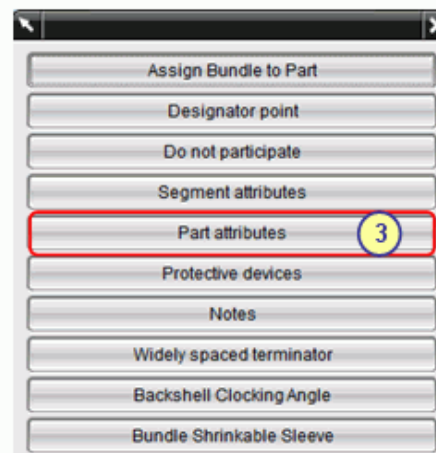
	1	These two lines represent together the wires path though the backshell. Line 1 is from the angle to the outer face where wires are connected.
	2	Line 2 is from the connector side to the angle of the backshell. These two lines share the same end point (at the angle).
	3	The optional multi-port is located at the extremity of Line 1.

Add Task Attribute to Rectangular angled backshell

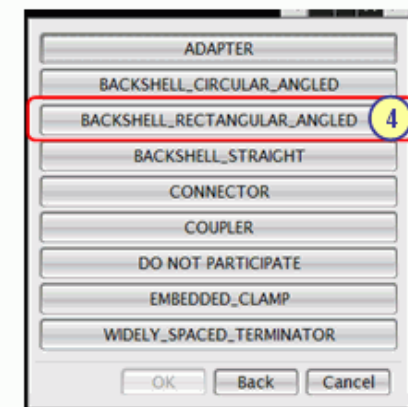
- ① Click on the  icon to open the "HarnesSYS Interface" window.
- ② Choose "Add task attributes", to open the **Add task attributes** window.
- ③ Press the "Part attributes" Option to open the **Part attributes** window.
- ④ In the window that appears, choose the **"BACKSHELL_RECTANGULAR_ANGLED"** attribute.



Main Interface window



Add task attributes window



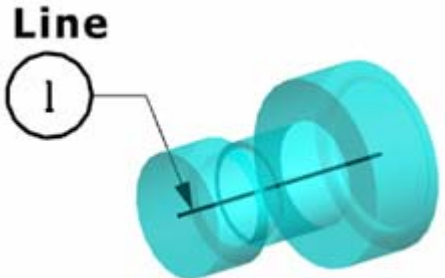
Part attributes window

Result: "HARNESS_TASK" = "BACKSHELL_RECTANGULAR_ANGLED" attribute is assigned to the NX part.

4.2.1.4 Adapter Structure

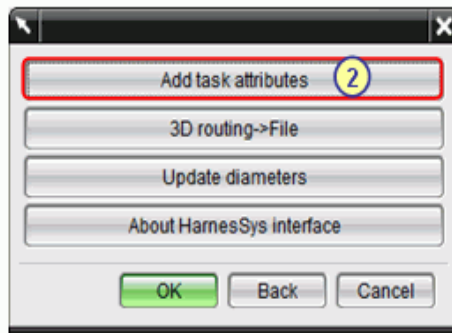
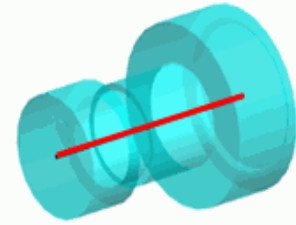
An **adapter** is defined by a NX Solid Body representing its geometry. In addition it should contain one line. Some adapters may include several wires paths.

Table 10: Representation of Adapter by NX objects

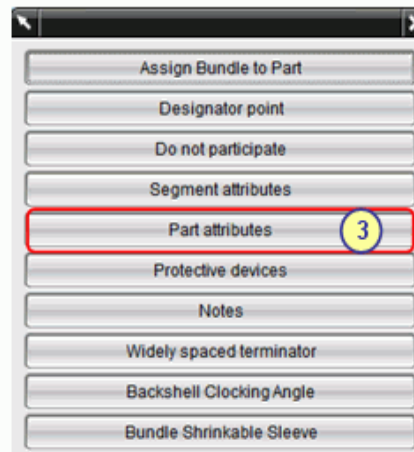
 Line 1	1	The line represents the wires path in the adapter connecting the designator part and the backshell. Its length should match the length of the wires path through the adapter. For adapters with several wires paths, there should be several non-collinear lines.
---	----------	---

4.2.1.5 Add Task Attribute to Adapter

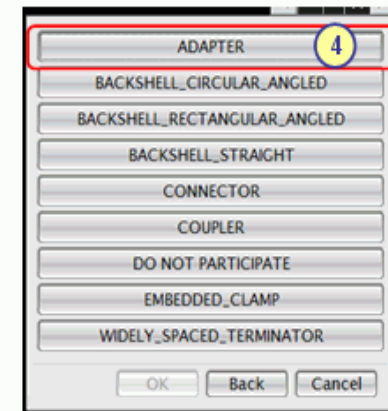
- ① Click on the  icon to open the "HarnesSYS Interface" window.
- ② Choose "Add task attributes", to open the **Add task attributes** window.
- ③ Press the "Part attributes" Option to open the **Part attributes** window.
- ④ In the window that appears, choose the "**ADAPTER**" attribute.



Main Interface window



Add task attributes window



Part attributes window

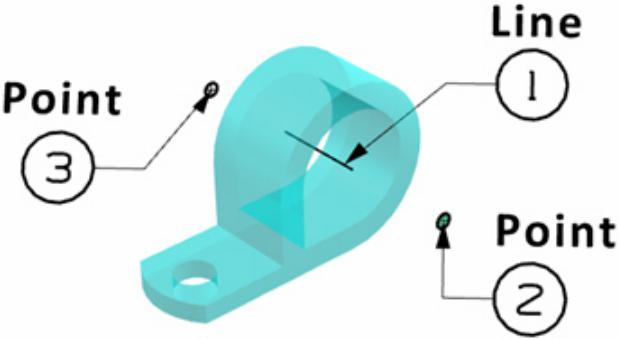
Result: "HARNESS_TASK" = "ADAPTER" attribute is assigned to the NX part.

4.2.1.6 Embedded Clamp Structure

Embedded Clamp Structure typically, **clamps** are not part of the cable assembly, they are part of the assembly structure. However, for compatibility with former versions of the interface, it is possible to embed a clamp in the cable assembly.

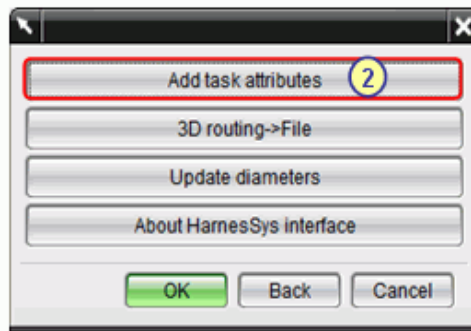
Each **Embedded clamp** is defined by a NX Solid Body representing its Geometry. In addition it should contain one line and two points.

Table 11: Representation of Embedded Clamp by NX objects

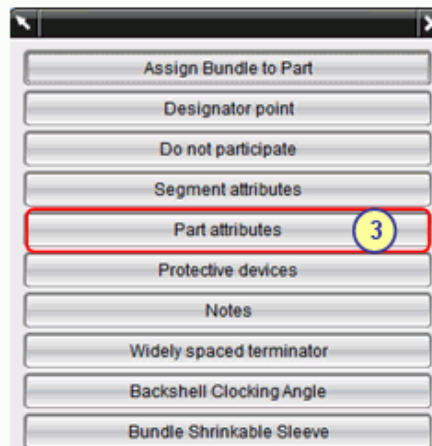
	1	The line describes the wires path in the Clamp. The length of the line defines the device width.
	2 3	Two points along the centerline path. These points indicate the straight portion of the wires crossing the clamp.

4.2.1.7 Add Task Attribute to Embedded Clamp

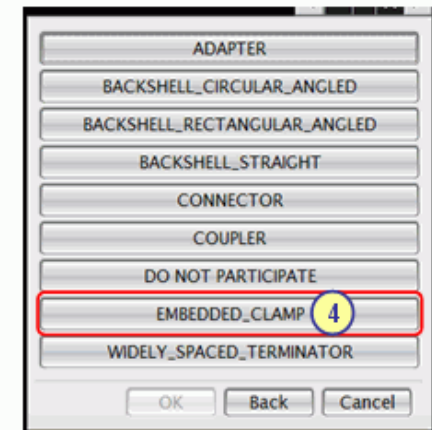
- ① Click on the  icon to open the "HarnesSYS Interface" window.
- ② Choose "Add task attributes", to open the **Add task attributes** window.
- ③ Press the "Part attributes" Option to open the **Part attributes** window.
- ④ In the window that appears, choose the "EMBEDDED_CLAMP" attribute.



Main Interface window



Add task attributes window



Part attributes window

Result: "HARNESS_TASK" = "EMBEDDED_CLAMP" attribute is assigned to the NX part.

4.3 Route the Geometrical Bundle

Route segments using standard as describe *in this manual* (on page 25). Follow NX Documentation on Electrical Routing Application and pay attention to the following items:

- a. **Route the MAIN Path** (on page 51)
- b. **Create Branch Segment** (on page 52)
- c. **Design Flexible Segments for 2D Manufacturing.** (on page 52)
- d. Do Not Participate uncompleted geometry.
- e. Special **end-connector** cases:
 - Use named points indicating wire end path. (on page 57)
 - **Route Paths for each Lug individually.** (on page 57)
 - **Assign Multiple Terminators at a Single Point** (on page 56).
 - **Widely Spaced Terminator on single part.** (on page 59)
 - **Data Bus Coupler.** (on page 61)

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[HarnesSYS - NX Interface Process.....](#)

4.3.1 Route the Main Path

The main path should contain the widest and longest branch segments. It should be routed before other branches. This is true even if the main path is curved as in the picture below.

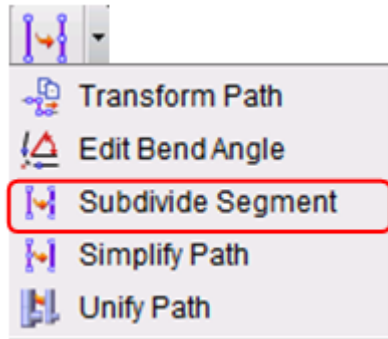
The reason for this is because it is drawn without a break on the JIG and such a break can not occur on the real model.



Figure 13: Route the Main Path

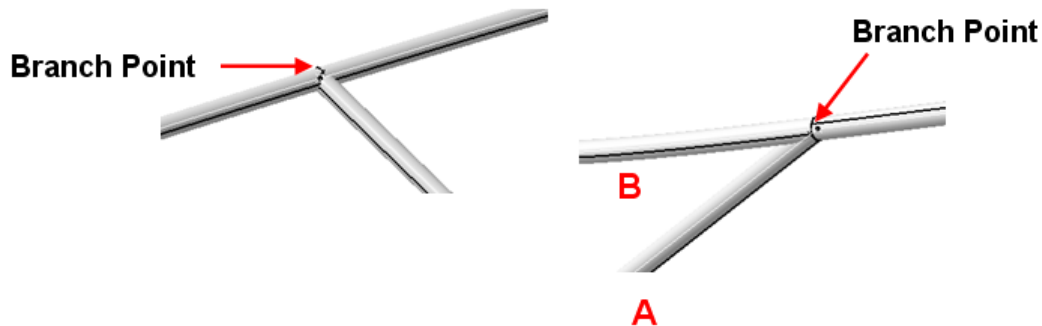
4.3.2 Create Branch Segment

In order to route a branch segment from the main segment, the main segment is subdivided using a Subdivide Segment NX option, and the second segment is routed from the created Routing Control Point.



The branch segment may be routed tangentially or normal to the main segment.

It is not recommended to route wires with a sharp angle, e.g. from point A to point B.



Tangential routing may occur when a main segment is split into several branches that are connected to different connectors on the same box.

4.3.3 Design Flexible Segments for 2D Manufacturing

The JIG diagram in **HarnesSYS™** for a flexible bundle is produced on a planar Jig table where as the routed branch segment is installed in 3D space. Therefore branch segment directions might have unexpected twisting angles during the actual installation. This situation may require adding length to branch segments and might cause unnecessary tension to the bundle supports.

However the routing is usually installed on planar or semi-planar surfaces like Skins, Frames or Sections. The user may see two such planar frames in the diagram below.

In this case it is recommended that when designing a new routing segment from an existing planar routing segment, it should start with a vector projected onto that planar surface.

In addition, it is not recommended to connect branches to segments in the area between two different planar surfaces. In the diagram below, you should not place branches in the designated



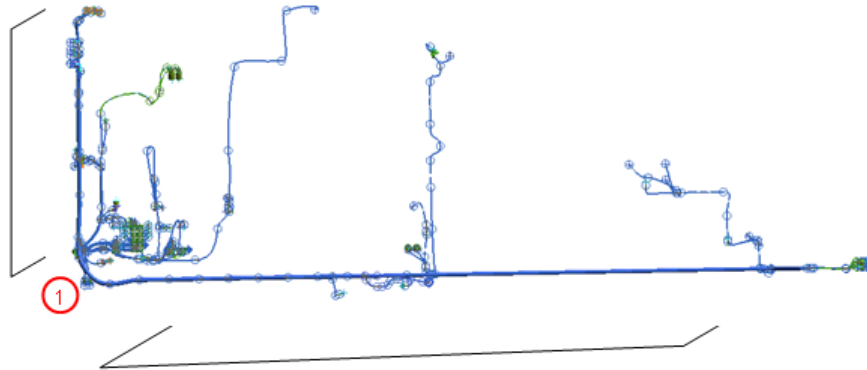


Figure 14: Two Planar Frames

4.3.4 Do Not Participate uncompleted geometry

"DO NOT PARTICIPATE" is an attribute that is set to NX object. Such NX object like NX parts, points or segments it will not be transferred to **HarnesSYS™**. This may be used for example when testing an alternative geometry and therefore there is no need to remove the extra data from the NX model. See definition details.

4.3.5 Special End Connector Cases

4.3.5.1 Route Paths for each Lug individually

Several lug may be installed on the same stud. In addition lugs may be installed o different studs or different pins of the same Reference Designator. In such cases wires for this Reference designator are routed separately creating a widely spaced terminator. The lug name that is set to each lug contains the Reference Designator name and may contain pin and lug ID according to the following format:

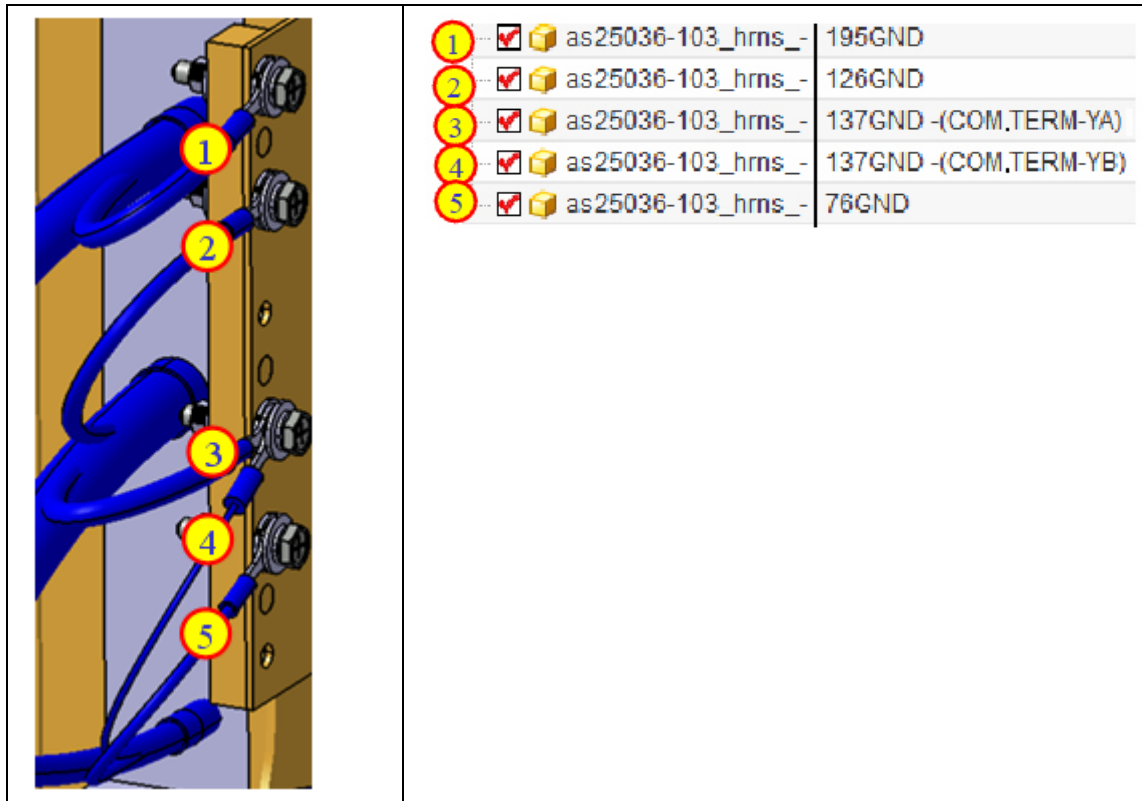
XXXX **YYY**(**ZZ**)

where:

XXXX - designator

YYY - optional pin name

ZZ - optional lug data.



Such Lugs are transferred from **HarnesSYS™** using wire data:

Designator, Pin Name, Term Code and Common Terminator Data.



One or more wires may be terminated on the same lug and several lugs may be installed on the same stud.



Route the path for each lug separately.

The following tabel indicates lugs combinations

Table 12: Example for lug naming conventions

No.	Cases	Lug in <i>HarnesSYS</i> ™	Lug in NX	
			Format	Example
1.	Ground (one stud)	- One lug per Reference Designator	xxxx	GND25
2.	Widely Spaced Terminator (several studs)	One lug per stud	xxxx yyy	TB23 A TB23 B
3.	Ground or Widely Spaced Terminator with Common Terminator	- Common Terminator - More than one lug per stud.	xxxx yyy(zz)	GND31 - (COM.TERM-YA) TB23 A (COM.TERM-YA)
4.	Ground or Widely Spaced Terminator: Several wires connected to the same stud without common terminator	- No Common Terminator implies one wire per lug - More than one lug per stud.	xxxx yyy(zz)	GND31 - (WIRE-505) TB23 A (WIRE-1001)

4.3.5.2 Assign Multiple Terminators at a Single Point

One wire path and only one backshell if present

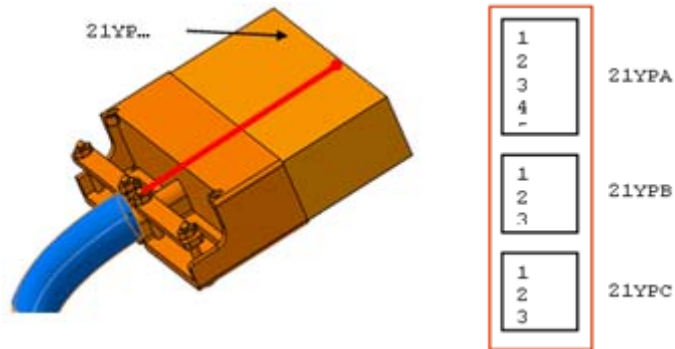
- Several inserts with pin numbering that may be repeated.
- Each insert has its own Reference Designator
- A single backshell for all the inserts

The connector is defined as "**Multiple terminators at a single point**".

In this case, several *HarnesSYS*™ Reference Designators may be mapped to a single NX wire termination point using wild card <...> which describes one wire path only.

Example

NX component name should be "21YP..." and it will be mapped to **HarnesSYS™** Reference Designators 21YPA, 21YPB and 21YPC.



The "Multiple Terminators at a Single Point" method may be also used for several circuit breakers inside a box or for other designators whose wires are routed and connected only after the bundle is produced.



The reference designator name of such Multiple terminator in **HarnesSYS™** contain asterisk for example "21YP*"

4.3.5.3 Use named points indicating wire end path

Designator Point indicates the end of a wire path and is used in the following cases:

1

When there is multi bundle designator it belongs to only one bundle in the NX model. The other bundles contain points referring to this designator.



Figure 15: Multi Bundle Designator

- 2 When it is a complex Reference Designator for example: a Widely Spaced Terminator , where there are number of pins on one NX part model.

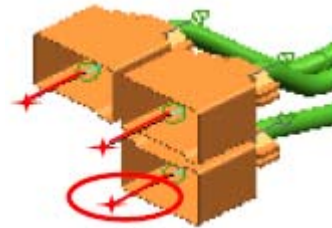


Figure 16: Connector Installation

- 3 When the NX model is not defined or not installed



Such a point (with the parameter HARNESS_TASK=DESIGNATOR_POINT) may also be used to temporarily represent a connector part, that even though the part is not yet defined in NX.

This point should be built at the bundle assembly level. It should contain a named with the **HarnesSYS™** Reference Designator Name. In addition its harness task is defined by object attribute. This object attribute is set to the point using the User Function program.

See definition details. (on page 78)

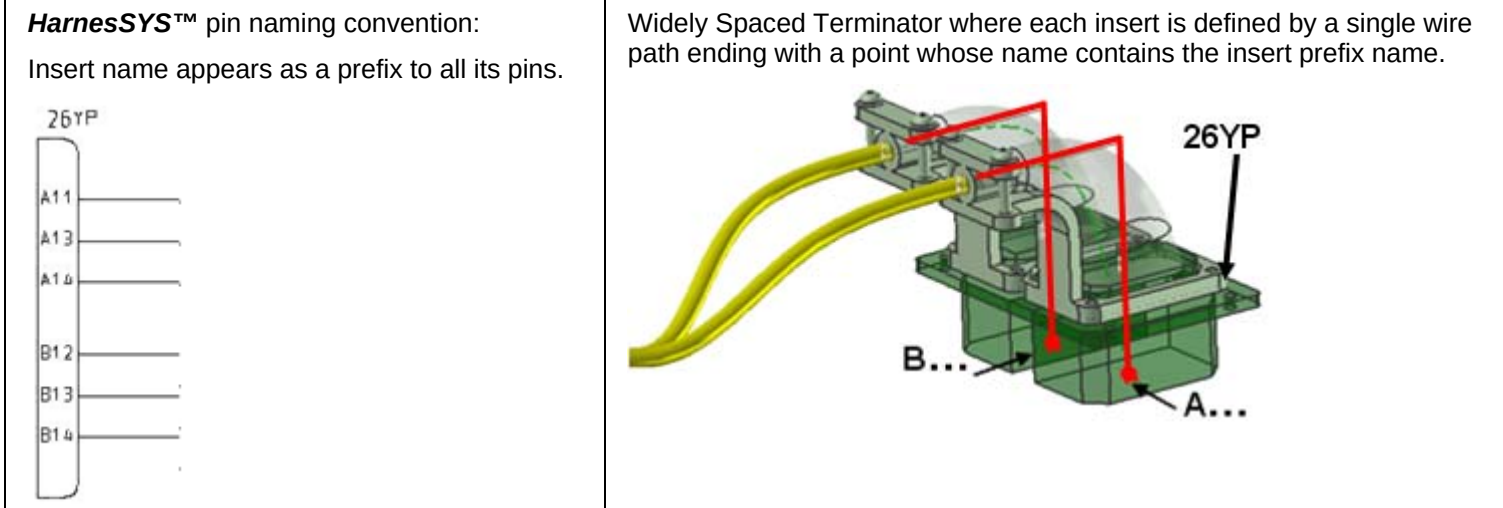
4.3.5.4 Widely Spaced Terminator on single part

Widely Spaced Terminator is a group of pin terminators targeted for a single connector that is widely spaced. The pin names, the lines and the part attributes appear in predefined parts. Each wire path may contain its own backshell.

Each of the terminators refers to the wires pin name in **HarnesSYS™**.

Example

In the following example inserts appear as a prefix of the Ref. Des. Pins



Alternatively, the following solution without Widely Spaced Terminator may be implemented if each insert is considered in **HarnesSYS™** as a separated Reference Designator.

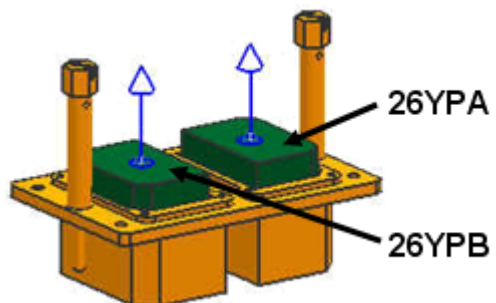
THarnesSYS™ pin naming convention: Pins are named independently for each insert.



Each insert is defined as a separate rectangular connector.



The Solid Body of the outer connector that contains the inserts does not participate in the bundle .



4.3.5.5 Data Bus Coupler

The **Coupler** Reference Designator is the interface name of the Data Bus coupler component. Data bus shielded group wires cable are connected to the Coupler using named points with predefined formats on each Coupler side. The ampersand behaves like or between two expression. "....." indicates wild cards.

In the example below, pin name with prefix 1 or 3 will be routed to the point with name "1...&3...". While wires whose pin name contain the prefix 2 or 4 will be routed to the point with name "2...&4...".

Example

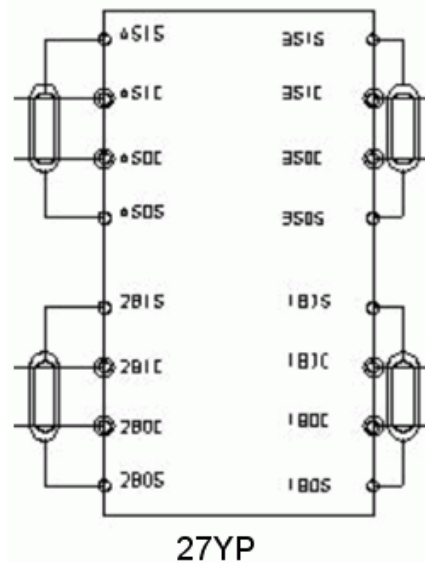


Figure 17: SWD Data Bus

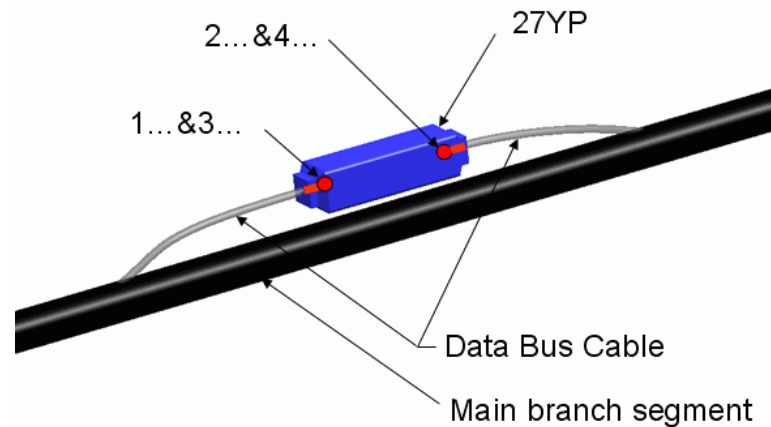
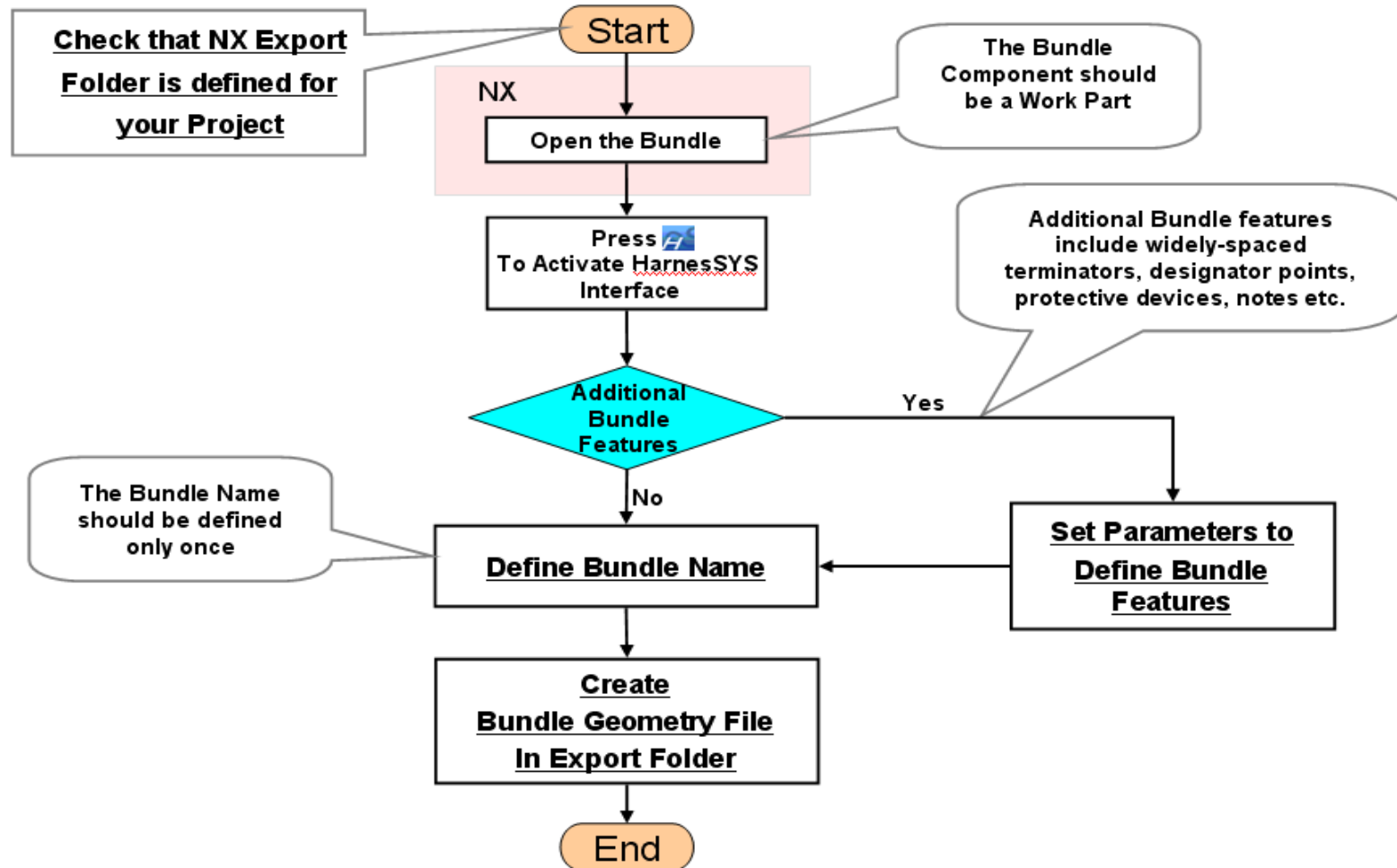


Figure 18: Data Bus Coupler Connections

4.4 Export Bundle Geometry

The User Function that is built inside NX module creates a file with the extension 3d asc at the specific predefined folder. The file contains the bundle segments geometrical definitional the User Function adds attributes to the bundle model and checks the validation of the NX geometry.

The following chart describes the work flow while transferring NX bundle model to **HarnesSYS™**.



 List of Links

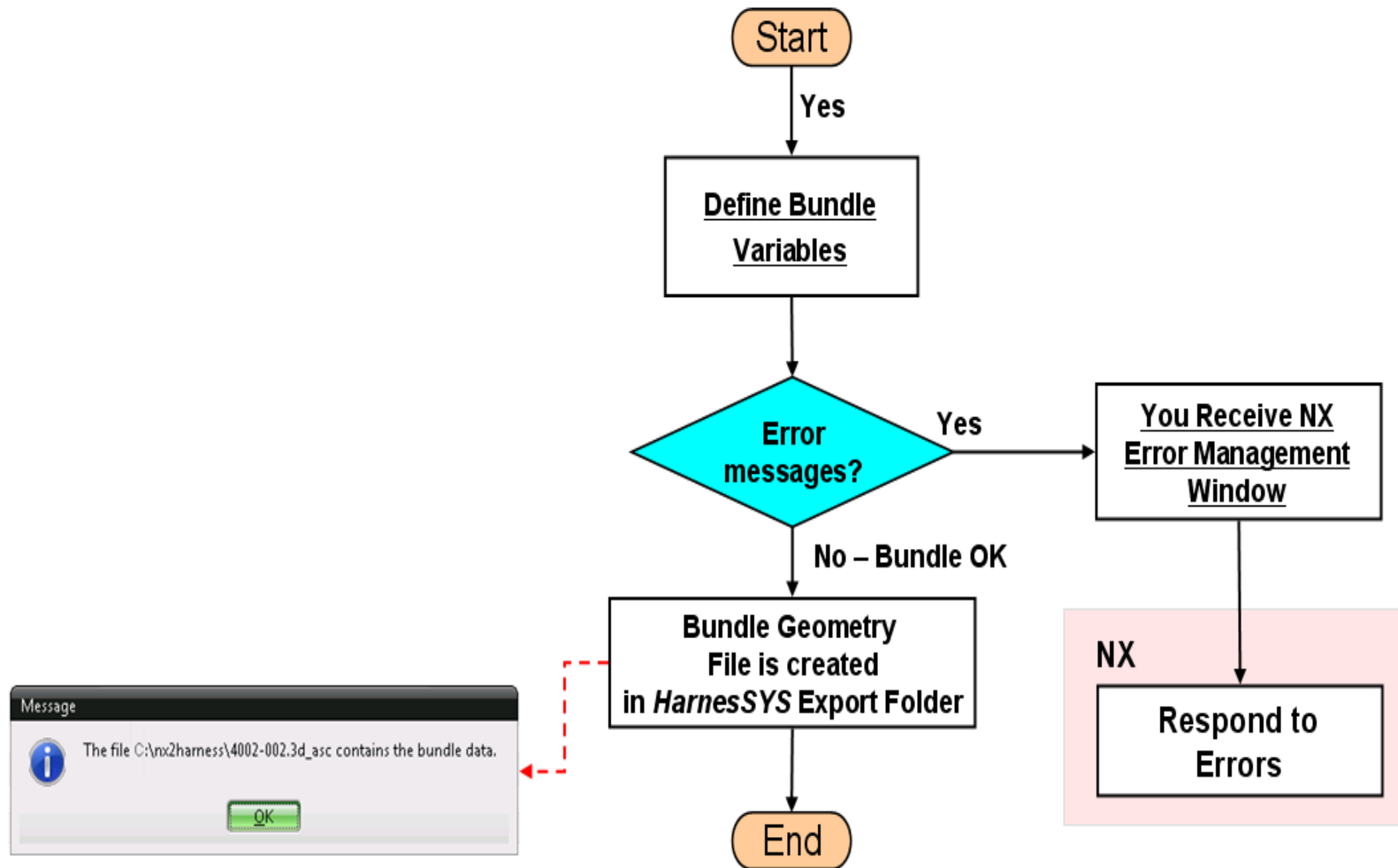
[Check HarnesSYS Import & Export Folder Definitions.....](#) 15

[Create Bundle Geometry File](#) 65

[Define Bundle Name Parameter.....](#) 70

[Set Parameters to Define Bundle Features.....](#) 73

4.4.1 Create Bundle Geometry File



 List of Links

Define Bundle Variables 67

NX Error Management Window 72

4.4.2 Define Bundle Variables

The screenshot shows a dialog box titled "Define Bundle Variables" with a close button (X) in the top right corner. The dialog contains ten input fields, each preceded by a yellow circle with a black number. The fields are as follows:

- 1 Through 4002-002.3d_asc
- 2 JIG BASE NUMBER 4002
- 3 JIG DASH NUMBER 002
- 4 Start path from 31P
- 5 End path at 36YP
- 6 Routing tolerance = 0.1
- 7 Angular tolerance = 1
- 8 Install tolerance = 1
- 9 Save model Yes
- 10 Confirm parameters

At the bottom of the dialog, there are three buttons: "OK" (highlighted in green), "Back", and "Cancel".

- 1 The variable holds the name of the file which contains the data to be transferred to **HarnesSys**. The default is xxxxx- yyyy. 3d_ asc, where xxxxx- yyyy (base and dash respectively) is the identification of the bundle.
- 2 The variable holds the bundle's base number as defined in **HarnesSys**.
- 3 The variable holds the dash of the BTP which identifies its effectivity range
- 4 The variable holds the initial Designator for the primary Jig path. Another part may be selected from a displayed list of all the Designators of this bundle.
- 5 The variable holds the Designator at the other end of the primary jig path. Another part may be selected from a displayed list of all the Designators of this bundle. See additional information below.
- 6 The variable indicates the maximum allowed distance between two curves for them to be considered connected, and the distance between a coordinate and curve when describing a curve using a collection of straight lines.

- 7 The variable indicates the maximum allowable angle between two consecutive chords when describing a curve using a collection of straight lines.

- 8 The variable indicates the maximum allowable distance between a point and a curve, (as described by a collection of straight lines), in order to consider the point as located on the curve. This variable must have a larger value than the route tolerance variable.

The 3d_asc file and the model contain new automatically generated segment and node names. It is important to save these names with the other geometric data to relate these to names in the HarnesSys JIG.

Choose the option:

- 9 1. **Save** - Save the model with automatically generated data (Recommended option, if the automatically generated data is not saved, the **Update diameters** option of the interface will fail to update the model)
2. **No** - No save of model with automatically generated data.
- 10 Clicking this option confirms all the other variables.

4.4.3 Define Bundle Name Parameter

When you start working with the **HarnesSYS™** Interface, the bundle-dash number must be assigned to the Bundle assembly part as **Part attribute** in order to recognize the **Bundle** within the bundle assembly.

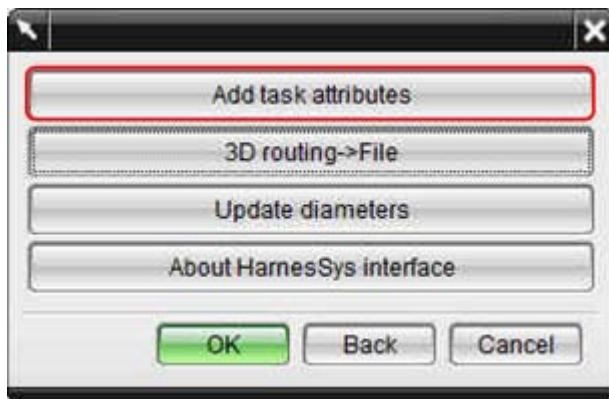
In order to assign the Bundle Name as a Part attribute of the Bundle assembly part, perform the following:



The Bundle assembly part should be the **Work Part**



Select the first activity from the **HarnesSYS™** Interface Main Window



The following "Categories of Electrical Tasks" menu is displayed:



In the "Categories of Electrical Tasks" menu that appears choose the "**Assign Bundle to Part**" Option and click **OK**.

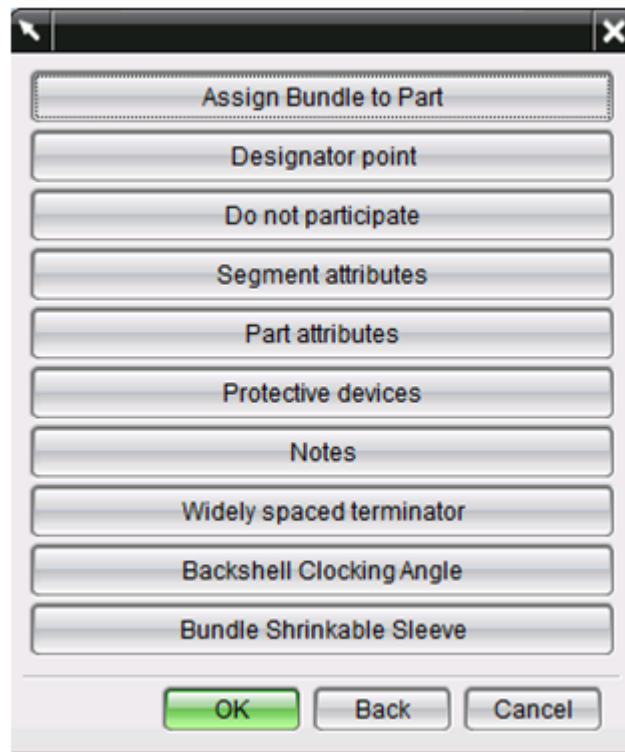


Figure 19: Categories of Electrical Tasks



The following pop-up menu appears



Type a value for the bundle name attribute with the format **xxxx-yyy** where :

xxxx is a bundle number

yyy is a dash number

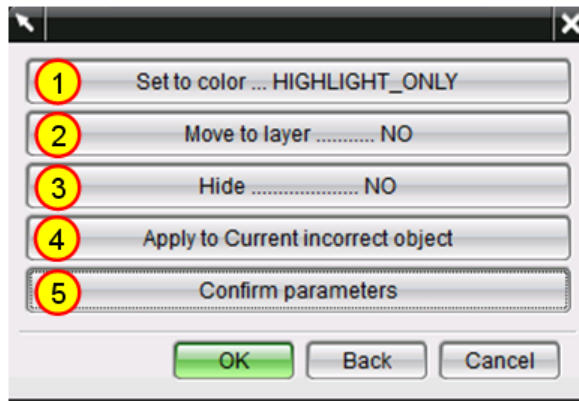
Example: 4002-501

⇒ Click OK to confirm attribute value.

To return to the **Main HarnesSYS™ Interface Window** choose **Cancel**.

4.4.4 NX Error Management Window

If some electrical data is absent or there is no continuity in the segment sequences, the error window will pop up to show an error diagnostics.



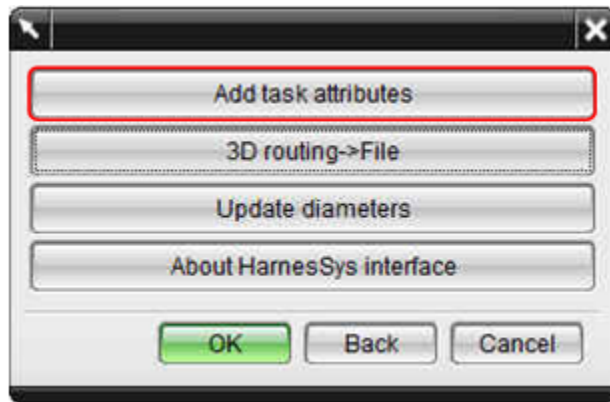
- ① Highlight the incorrect object with the chosen color. Contains the highlight option also.
- ② Move the incorrect objects to the desired layer [1-255].
- ③ Move the incorrect object to blank.
- ④ If this option is activated, the above changes will occur in the current incorrect object and the menu will appear again if another incorrect object is detected.
- ⑤ If this option is activated, the above changes will occur in all incorrect objects without any other prompts.

Figure 20: Error Window

4.4.5 Set Parameters to Define Bundle Features

4.4.5.1 Add Task Attributes Option

Select the first activity from the **HarnesSYS™** Interface Main Window. This activity sets the specific attributes of the NX model before the Bundle Geometry file (*.3d_asc) is created.



There are two kinds of **HarnesSYS™** Attributes:

1. **Part attributes** - Assigned to an NX Work Part
2. **Object attributes** - Assigned to an NX Object
 - When **Part attributes** are assigned to the **Bundle** (the top-assembly part), it should be the **Work** and **Displayed Part**.

When **Part attributes** are assigned to a **Bundle component**, the component part should be the Work Part.

- When **Object attributes** are assigned to an **NX-Object (line, point etc.)**, the **Bundle object** should be the **Display Part** and the owning Component should be set as the **Work Part**.



The following "Categories of Electrical Tasks" menu displayed:

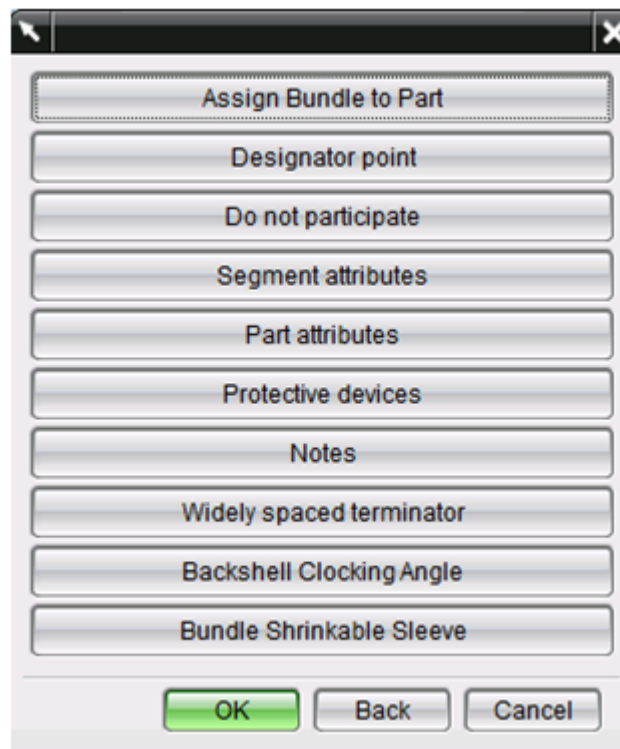


Figure 21: Categories of Electrical Tasks Menu

The following List of Links enable you to jump to appropriate place in this user manual.

 List of Links

- 1. [Assign Bundle Number](#) 78
- 2. [Designator Point Definition](#)..... 78
- 3. [" Do not participate" Definition](#) 78
- 4. [Segment Attributes](#)..... 79
- 5. [Part Attributes](#) 81
- 6. [Protective devices](#)..... 82
- 7. [Notes](#)..... 83
- 8. [Grouping Widely Spaced Terminators](#) 85
- 9. [Backshell Clocking Angle](#)..... 81
- 10. [Bundle Shrinkable Sleeve](#) 85

Add task Attributes Option (Continue)

Table 13: Categories of Electrical Tasks

Category	Description
Assign Bundle to Part	Enables the user to assign the HARNESS_BUNDLE_NAME =bundle-dash Part attribute to the top-assembly Work and Display part.  When starting work with the HarnesSYS™ Interface, the bundle-dash number must be assigned to the top-assembly as a Part attribute in order to recognize the Bundle within the bundle assembly.
Designator Point	The attribute with "DESIGNATOR_POINT" value should be assigned to the termination point in the devices such as connector, WST etc. The attribute with "DESIGNATOR_POINT" value may also be assigned to a point on the routing object in the bundle, indicating a termination of the routing in the aircraft. See detail (on page 57)
Do not Participate	Enables the user to assign a HARNESS_TASK that implies ignoring the selected component or object. See detail
Segment attributes	Enables the user to assign attributes to a Routing Segment .
Part attributes	Enables the user to assign HARNESS_TASK attributes as Part attributes . This function is used to define components in a part library. In order to use this function the user must set the desired part to be the current Display Part . See detail (on page 26)
Protective devices	Enables the user to assign attributes to the protective devices.
Notes	Enables the user to assign notes such as Red Label, Clamp Marker etc. as Object attributes.
Widely spaced terminator	This obsolete option Enables the user to assign the HARNESS_TASK = WIDELY_SPACED_TERMINATOR attribute to NX group object. Widely Spaced Terminator may be defined within a single part using Part attribute. Alternatively several lugs may define Widely Spaced Terminator using naming convention to each individual lug.

HarnesSYS - NX Interface Process

Category	Description
Backshell Clocking Angle	Enables the user to set the keyway direction of the backshell as a line attribute. See detail
Bundle Shrinkable Sleeve	Enables the user to assign larger diameters in NX than they are in HarnesSYS™ (this may occur appear when a bundle is sealed in a fuel tank)

Assign Bundle Number

A bundle number is set as a Part attribute in the NX Part. It indicates that the NX Part and its Component Parts contain this bundle information.

 Select "**Assign Bundle to Part**" Option in the "Categories of Electrical Tasks menu" and click **OK**. The following pop-up menu appears



Figure 22: Cue: Enter HARNESS_BUNDLE_NAME value

 Type a value for the bundle name attribute with format xxxx-yyy where :

- xxxx is a bundle number
- yyy is a dash number

Example name attribute: 4002-501

 Click OK to confirm attribute value.

To return to the **Main HarnesSYS™ Interface Window** click **Cancel** again.

Designator Point Definition

See usage details.

To assign a Designator Point attribute:

 Select the "Designator Point" option in the "Categories of Electrical Tasks menu" and click OK.

The regular SELECT MENU of NX is displayed.

 Select a point and click OK .

To return to the Main Tasks menu choose Cancel.

To return to the Main **HarnesSYS™** Interface Window choose Cancel again.

" Do not participate" Definition

See usage details.

These object attributes may be removed using the regular NX tools. To assign a "Do not participate" attribute perform the following:



Select the "Do not participate" option in the "Categories of Electrical Tasks menu" and click OK .

The regular SELECT MENU of NX is displayed.



You may select a Component, Group of objects or a Routing object.

- To return to the **Main Tasks menu** choose **Cancel**.
- To return to the **Main HarnesSYS™ Interface Window** choose **Cancel** again

Segment Attributes

Segment attributes such as EMI or RUN letter will be shown on the **HarnesSYS™** jig (form board) Braid thickness will set additional segment thickness according to the segment diameters in **HarnesSYS™**.

To assign attributes to Segment Curves:



Select the "**Segment attributes**" option in the "Categories of Electrical Tasks" menu.



Click **OK**.



The Harness Tasks for Segment Attributes menu appears.



Figure 23: Cue: Enter 'HARNESS_CLAMP_MARKER' value



This category contains the assignment of different curve types.



Select one of the following options:

1. HARNESS_BRAID_THICKNESS.
2. HARNESS_EMI, HARNESS_EMI_GROUP_NAME & HARNESS_RUN_LETTERS.

HARNESS_BRAID_THICKNESS

To assign the sleeve to the appropriate Segment/Curve:



In the pop-up window, enter the sleeve thickness:

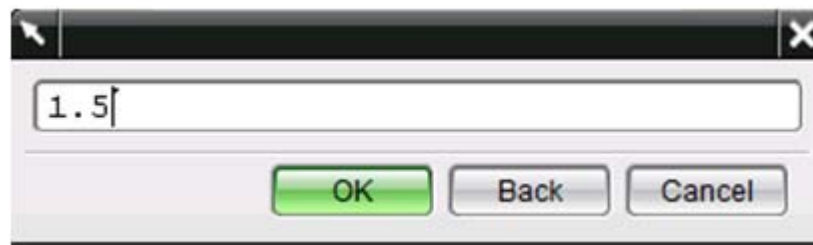


Figure 24: Cue: Enter HARNESS_BRAID_THICKNESS value

The regular SELECT MENU of NX is displayed.



Choose the segment to which to add the sleeve value to the segment diameter of the single segment, or choose the bundle object to which to add the sleeve value to the diameter of all segments in the bundle object.

- To return to the **Main Tasks menu** click **Cancel**.

- To return to the **Main HarnesSYS™ Interface Window** click **Cancel** again.

HARNESS_EMI, HARNESS_EMI_GROUP_NAME & HARNESS_RUN_LETTERS

These are alternative attributes that may be set on segments. They add information about the EMI level applicable to that segment.

To assign Run Letters, Harness EMI or Harness EMI Group Name to segments, select one of the attributes:

HARNESS_EMI , HARNESS_EMI_GROUP_NAME , HARNESS_RUN_LETTERS
from the "Categories of Electrical Tasks" menu and click **OK**.

The following pop-up window appears (not for HARNESS_EMI):

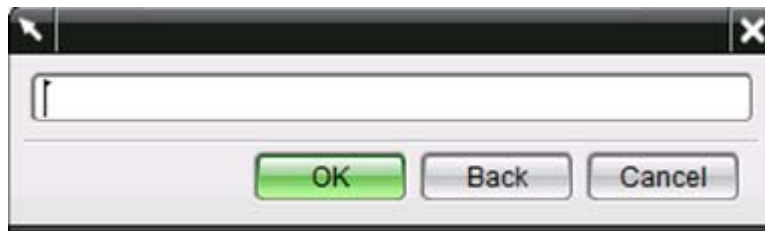


Figure 25: Cue: Enter EMI_GROUP_NAME (HARNESS_RUN_LETTERS) value.

Enter the Name and press OK.

The regular SELECT MENU of NX is displayed. The User may select centerline objects such as lines or Segment Curves, which represent segments with **HARNESS_EMI** , RUN LETTERS or EMI GROUP NAME.

- To return to the **Main Tasks menu** click **Cancel**.
- To return to the **Main HarnesSYS™ Interface Window** click **Cancel** again.

Part Attributes

Add Task Attribute to Basic connector (on page 31) 

Circular Connector Structure (on page 32) 

Add Task Attribute to Data Bus Coupler (on page 35) 

Add Task Attribute to Rectangular connector (on page 37) 

Add Task Attribute to Widely spaced terminator on single part (on page 39) 

Backshell Clocking Angle

See usage details.

To assign an attribute to the keyway Line:



Select **Backshell Clocking Angle** option in the "Categories of electrical tasks" menu.



Click **OK**.



The regular SELECT MENU of NX is displayed.



Select the keyway line object and click **OK**.

- To return to the **Main Tasks menu** click **Cancel**.
- To return to the **Main HarnesSYS™ Interface Window** click **Cancel** again.

Protective devices

To assign attributes of Protective Devices, select the "**Protective devices**" option in the "Categories of electrical tasks" menu and click **OK**.

The **HarnesSYS™** Tasks for Protective Devices menu appears.

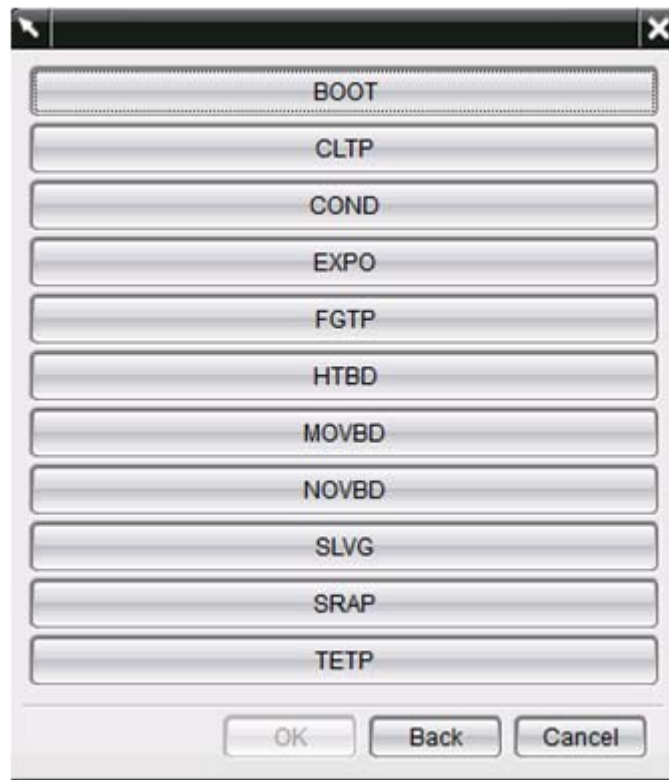


Figure 26: HarnesSYS™ Tasks for Protective Devices Menu



Select one of the different types of protective devices.



The regular SELECT MENU of NX is displayed.



The user may select Component or Group of objects. Each of them collects all the edge points of each protective device on the segments.

- To return to the **Main Tasks menu** click **Cancel**.
- To return to the **Main HarnesSYS™ Interface Window** click **Cancel** again.

Notes

To assign a note attribute, select "**Notes**" option in the "Categories of electrical tasks" menu and click **OK**.

The Harness Tasks for Notes menu appears:

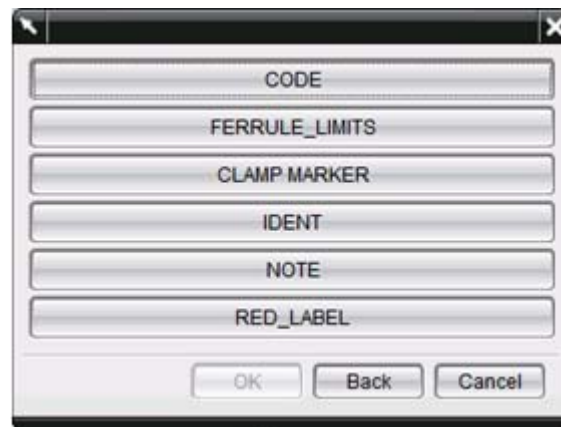


Figure 27: Harness Tasks for Notes Menu



Select one of the different types of notes.



To assign a Clamp marker Name to a point, select: CLAMP MARKER



The following pop-up window appears:

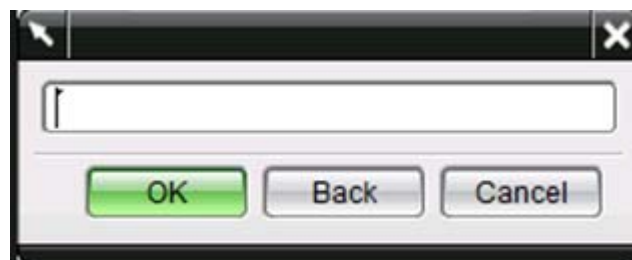


Figure 28: Cue: Enter 'HARNESS_CLAMP_MARKER' value

The regular SELECT MENU of NX is displayed.

The user may select a Point.



To assign any other note, select: CODE or FERRULE_LIMITS or IDENT or NOTE or RED_LABEL

The regular SELECT MENU of NX is displayed.

The user may select a Point.

- To return to the *Main Tasks menu* choose **Cancel**.

- To return to the **Main HarnesSYS™ Interface Window** choose **Cancel** again.

Grouping Widely Spaced Terminators

This obsolete option is used to set attribute to a grouping object for several connector that indicates wire ends in each pin separately.



Lug is no longer defined as a WIDELY SPACED TERMINATOR object attribute.

See [Route Paths for each Lug connected to the same Stud](#) 

Bundle Shrinkable Sleeve

To assign the same thickness to all segments in the bundle:



The attribute is added to the Display Part.



Select the "Bundle Shrinkable Sleeve" option.



In the interface window that appears enter the Braid thickness.

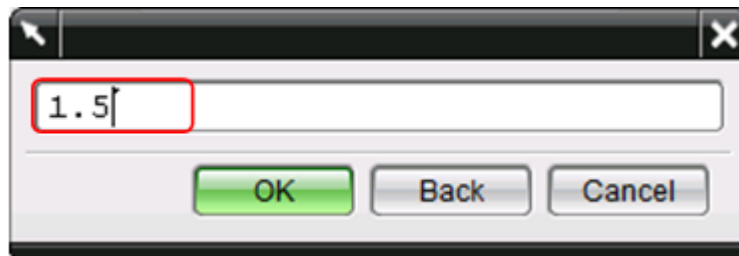


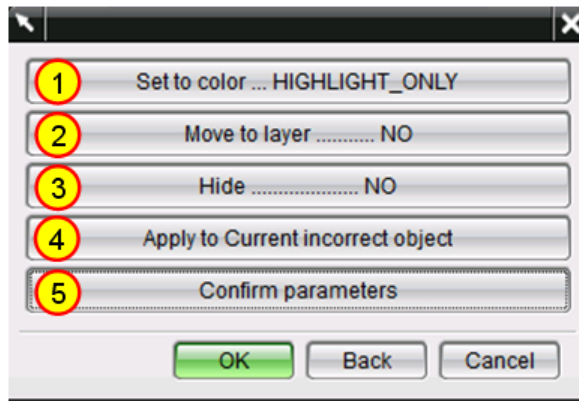
Figure 29: Cue: Enter 'HARNESS_BUNDLE_BRAID_THICHNESS' value



Click OK.

4.4.6 Error Management Window

If some electrical data is absent or there is no continuity in the segment sequences, the error window will pop up to show an error diagnostics.



- 1 Highlight the incorrect object with the chosen color. Contains the highlight option also.
- 2 Move the incorrect objects to the desired layer [1-255].
- 3 Move the incorrect object to blank.
- 4 If this option is activated, the above changes will occur in the current incorrect object and the menu will appear again if another incorrect object is detected.
- 5 If this option is activated, the above changes will occur in all incorrect objects without any other prompts.

Figure 30: Error Window

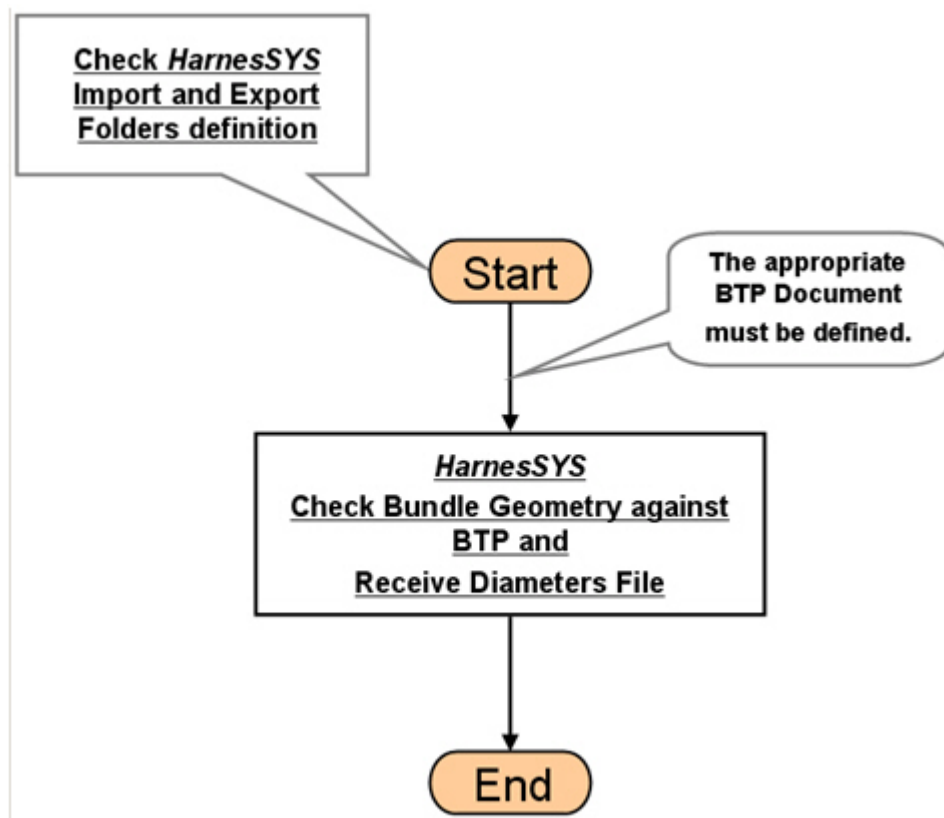
4.5 Check Bundle Geometry against BTP & Receive Diameters

This function compares Wire List and Part List data appearing in the BTP document with the bundle geometry data of the 3D model in XML format.

In This Chapter:

[HarnessSYS: Check Bundle Geometry against BTP](#) 88

The following scheme describes the *HarnessSYS*[™] requirement prior to this activity.



List of Links

[Check HarnessSYS Import & Export Folder Definitions](#) 15
[HarnessSYS: Check Bundle Geometry against BTP](#) 88

4.5.1 HarnesSYS: Check Bundle Geometry against BTP

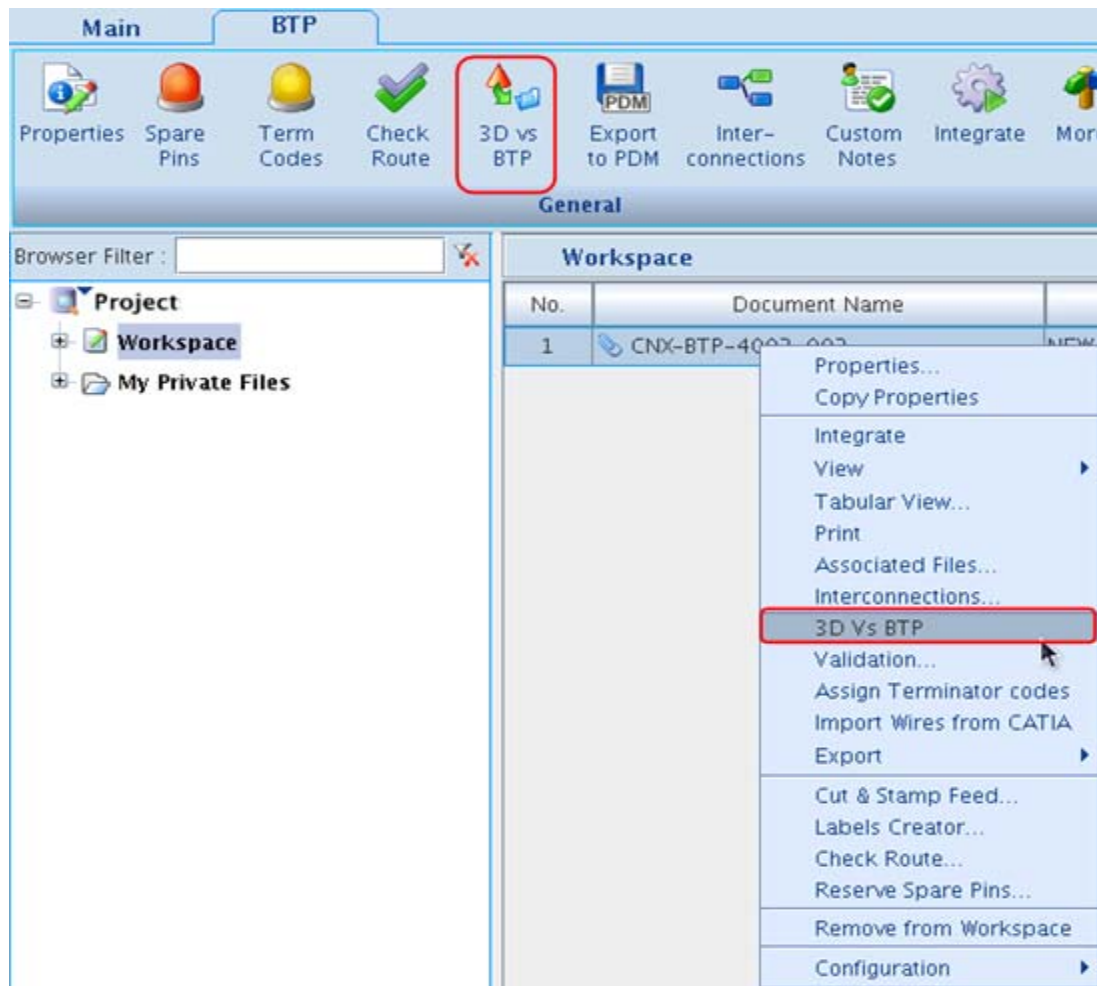
This function compares Wire List and Part List data appearing in the BTP document with the bundle geometry data of the 3D model, in XML format.



Select the **appropriate BTP document** from the *HarnesSYS™* workspace.

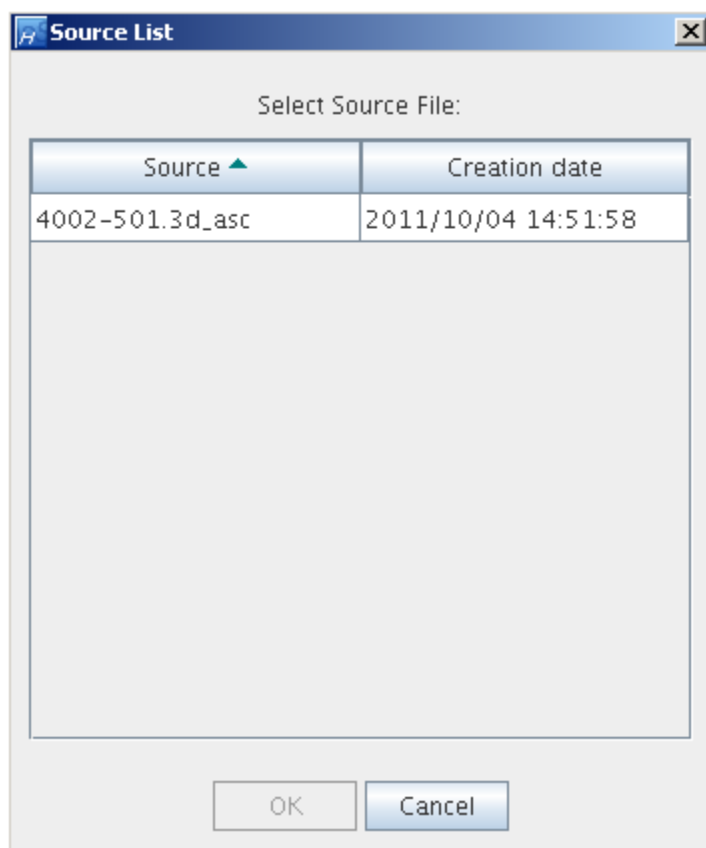


→ Press **MB3** → **3D Vs BTP** or choose the "3D vs. BTP" icon from the **BTP** toolbar.





The following menu appears:



It contains the 3d_asc files that appear in the **THarnesSYS™ import folder** and have the same BASE NUMBER as the BTP base number.



Select the desired file and click OK.

The check process is run in another window. The check process is similar to the Integrate BTP process described in "**Integrate BTP**" user manual.

The following log file shows a typical result. The notes indicate that this check process indents to NX users in order to check their data against the **HarnesSYS™** model.

- Base-dash number value was assigned as an attribute to the Bundle using the **HarnesSYS™** - NX interface.

Text File CNX / CNX-BTP-4002-002.log

```

*** Diagnostics for application CHECK-DIAM; Created at 2012/03/12 10:38:00 ***

PROJECT = CNX
DTP DOCUMENT = CNX-DTP-4002-002

1 The new source "/userhome/harness/ug2/harness/4002-002.3d.asc" for the JIG had been created at "2012/03/12 08:24:02"

WARNING (JG-0235):
The following PARTS are in the WIRE_LIST and do not exist in the GEOMETRY
PART = GND222
PART = PF002
PART = PI02

WARNING (WL-5202):
Part number 'D38999-20W-25_HRNS' in the 3D model doesn't match the Part Spec name in the database 'D38999/20WJ35PN' for designator 'J1005 (occ. key 252)'

2 Data exists in file '/userhome/harness/harness2ug/4002-002.3d.asc '

The CHECK-DIAM process has finished with warning(s)

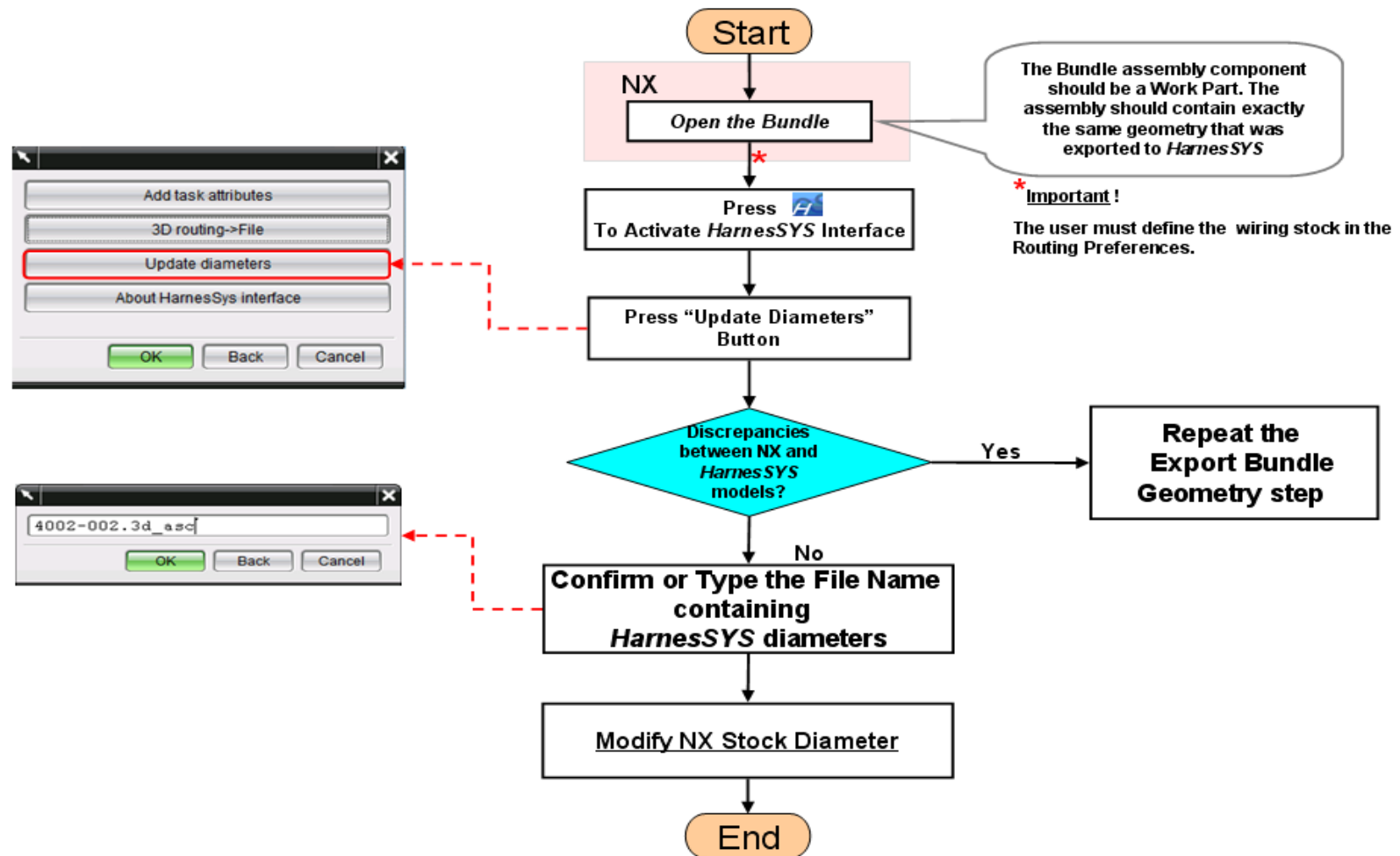
The following files are in your PRIVATE directory:
Errors listed by severities and codes : CSN-BTP-4002-002.err
Log file (screen messages)           : CSN-BTP-4002-002.log

3 The CHECK-DIAM process is finished successfully.

THE PROCESS TERMINATED AT 2012/03/12 10:38:02
    
```

- 1 There is a mismatch between parts: straight backshell in *HarnesSYS* and angled backshell in 3D model
- 2 *HarnesSYS* saves another 3d_asc file in the *HarnesSYS* Export folder appearing in the project default with the diameter data of each *HarnesSYS* segment. This data may be transferred to NX.
- 3 There is no commit to the database since nothing has been changed in *HarnesSYS*. All the results appear in files.

4.6 Update HarnesSYS Diameters



4.6.1 Display or Modify NX Stock Diameter

Load To Workspace

Application: All Sub Type:
 Status: All Base:
 Name:
 Effectivity:

Model	Block	From	COB	To

 From Date: To Date: 2012-11-22
 Authority:
 Search Clear Close

Information
 2012/11/27 10:1

Effectivity:

Model	Block	From

 From Date: To Date: 201
 Authority:
 Search

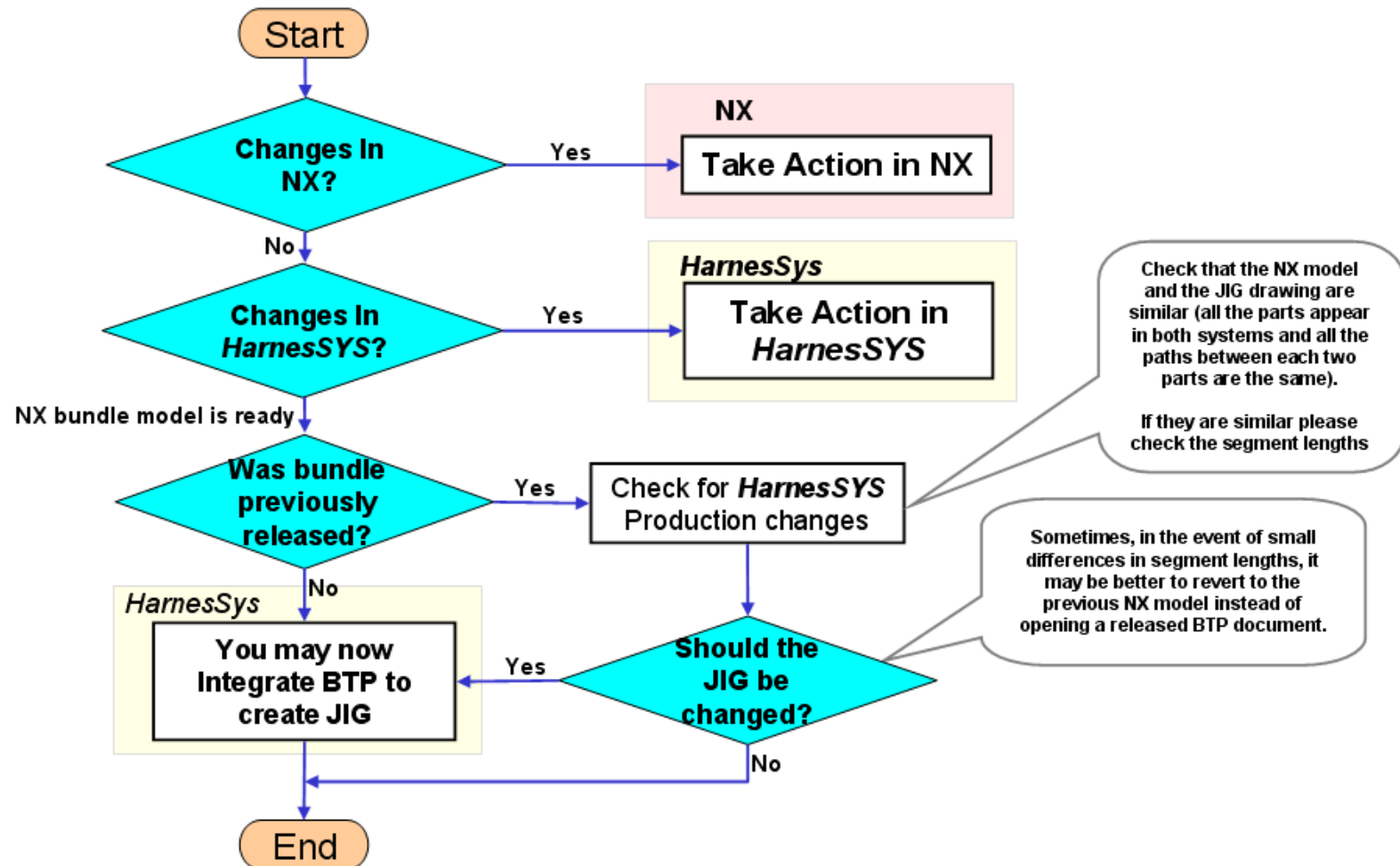
November 2012

Sun	Mon	Tue	Wed	Thu	Fri	Sat
				1	2	3
4	5	6	7	8	9	10
11	12	13	14	15	16	17
18	19	20	21	22	23	24
25	26	27	28	29	30	

Today is 22 November 2012

4.7 Apply the Bundle Changes

It is good practice to consult Electrical Designers before extensive changes are applied, because the bundle is designed by different disciplines. Sometimes changes in *HarnesSYS*™ may provide a better solution.



4.8 Integrate BTP to Create Jig

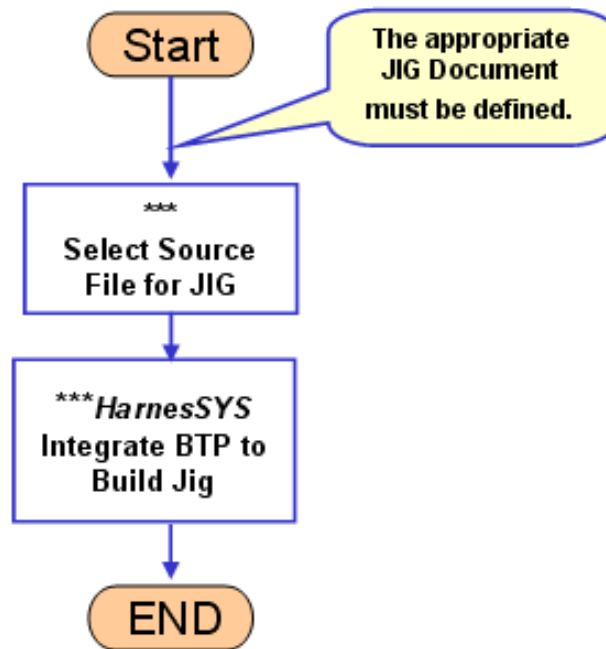


Figure 31: Integrate BTP to Create Jig Scheme

*** BTP processes are defined in detail in the **HarnesSYS™** Integrate BTP User Manual. The Integrate BTP process includes functionalities such as:

- Build Jig
- Build Not-to-Scale 2D Drawing
- Update Existing Jig
- Update Wires Lengths

4.9 Confirm BTP and Release



*** This process is defined in detail in the *HarnesSYS*[™] Integrate BTP User Manual.

 [Home](#)

[Introduction](#)..... 1

Glossary of Terms

A

Abbreviation Class

Defines the class of part (for example P for plug, SW for switch, etc)

Access Door

Access point (flap, door or other entrance) in aircraft used to install or access the part or bundle.

Area Code

The area code indicates the physical location of the part in the production unit.

Authority

Documents that authorize changes. They are located in the Authority Library

B

Backfield

Protection for connector which provides grounding or mechanical support

Box Interface

Boxes are independent electrical units, often connected to bundles by specific interface.

Braid Stop

The distance between the BRD and the end of the braid (see Project Defaults, Manufacturing).

Branch segment (branch)

Is a geometric JIG element containing wires. The branch is identified by uniformity of its wires.

BRD

Basic Reference Designator, also referred to simply as a Designator, the connector's local name.

BTP (Build-To-Package)

Document describing the packaging to produce a bundle. Generally it is a collection of WL/PL and JIG DATA. Must contain at least WL or PL.

Bundle

Each wire must belong to a bundle. A bundle name is composed of a bundle (base) number and dash number. The number of the bundle to which the wire belongs is often entered before the wire name on the SWD.

C**CAD**

Computer Aided Design

Contact Key

Code determining number and names of pins, defined in Contact Configuration Library. A part number has a unique contact key.

Contact Type

Type of contact: 4 options – Pin, Stud, Socket, Solder

D**Dash Number**

A descriptor of a document or bundle which associates the document with effectivity.

Designator

Each occurrence of a part in a SWD has a "code name" e.g. YP100. This is the (base) designator of the part. This number is unique in the Product.

Device Type

Type of device installed in bundle. In *HarnesSYS™* there are 15 options.

Diameter

The diameter of the segment which is calculated using a fixed formula which takes into account the various types of wires that pass through the segment.

Document

Configuration controlled document of a specific type such as SWD/WL/PL/BTP.

E**Effectivity**

The effectivity of an item is the set of production units (i.e. the subset of all the production units being manufactured) to which that item applies.

The following documents have effectivities: part list (with dash number), wire list (with dash number), SWD (with dash number).

The following items also have effectivities: wire, part occurrence, bundle (with dash number). In effect the effectivity of a bundle (with dash number) is the same as the effectivity of its wire list (with dash number).

EGN

An attribute assigned to segments to enable separation of wires with different EMI attributes.

EMI

Electromagnetic interference. The code value representing the susceptibility to interference with the signals that a wire carries when these signals are disrupted by electromagnetic emission from adjoining current carrying wires.

F**Family**

Family code ferrule connecting several shields, name fields in wire record, Find Base Number Part List W/L Dash flt file FROM_WIRE_INTEGRATE.

I**Integrate BTP**

When the project default parameter **Multi Path allowed** is not checked, an error is displayed if the integrated source contains Multi Paths.

Integrating a Wire

Integration of part integration process data flow logic flow intersecting effectivity issue.

J**Jig**

A full scale 1:1 layout for the construction of a wire bundle. Also called Form-board.

Jumper

Name number translation of a short default wire.

L**LCD**

last Change Date. Date document last changed.

M**Mating Connector**

The matching Connector assigned to a connector via the Part Occurrence mating field.

Multi Path

A path or a part of a path, which includes more than one alternative configuration for routing wires.

N**Null Path**

The default path, if there is no correlation between segments and wires. If there are several alternative routing options at a node and there is not enough information to define the path, the null path should be the longest one.

P**Parts List (PL)**

List of parts contain in a specific version of a bundle.

Path

Is a collection of continuous branch segments for routing wires. There may be single path or several parallel paths between two designators.

Plug

A device which by insertion to a receptacle establishes contact.

Plug Program Number (PPN)

The PPN (Plug Program Number) of a part is a key determining the order in which wires should be attached to the pins of the part. The key has 3 fields: **major, sub and degree.**

Private File

A file, generally a report, that is produced by a **HarnesSYS™** transaction, stored in user's private directory and only available to the user initiating the process.

Production Unit

The unit being designed or manufactured e.g. an aircraft.

Protective Device

A lamination protecting the bundle wires, such as; boot, sleeving, conduit or spiral-wrap.

R**Receptacle**

A contact device connected to a power supply and equipped to receive a plug.

Retrofit Installation Drawings- RID

Drawings showing only the change.

Retrofit Master Document RMD

Folder that contains all the documents required to implement the retrofit

Revision Letter

Letter determining the issue of a document. Every time a document is opened in order to modify it, its revision letter is raised. It can have the values **NEW** or **A, B, C** etc.

In addition the revision letter may include point release data. Point release is sequential number indicating inner release of the bundle. The document may be released in the following order:

00.01	00.02	NEW	NEW.01	NEW.02	A	A.01	A.02... etc
Point releases		Firm release	Point releases		Firm releases		

Routing

The path taken by a wire from one part to another.

S**Same Wire**

In 2 (SWD) System Wiring Diagram sequence number shield levels name significant wire name spare pin special designators splice named translation of unnamed spt file starting/ending point status suffix SWD symbol bank System wiring Diagram.

Segment (Branch)

A section of a bundle that lies between two nodes or between a part and a node or between two parts. Also referred to as a branch.

Segment Length

The length of a segment or branch, stated in millimeters or inches as defined in the Project Defaults. This length can be between two nodes or between a part and a node or between two parts.

Service Code

Type of retrofit change: A (Add), D (Delete), C (Change), B (Before).

Shell Code

Shell codes contain information about connector back shells (or rear side of the connector). The three variables concerning shell codes are the size, whether the connector is a receptacle or a plug, and whether it is straight or angled. Thus each shell size can have four shell codes.

Shrink Ratio

The nominal ratio of expanded diameter to recovered diameter of heat shrinkable tubing

Signal Type

Type of signal at interface box connection – 13 options:

Communication, GND Analog, GND Digital, GND Chassis, GND Power DC, GND Power AC, Lighting, MUX-BUS, Power AC, Power DC, RF, Signal Analog, Signal Digital

Sleeve Size

Diameter of sleeve around wire

Spare Pin

Is a designator pin, left unused (unattached to a wire). The pin requires special treatment such as capping.

Stick Diagram

Initial Jig diagram where all the lines are straight, the lengths of the branches are drawn to scale, and the dimension lines are removed.

System Wiring Diagram (SWD)

Old names: Schematic Diagram or Wire Diagram or Wiring Diagram.

A document which presents the design of a logical system e.g. air conditioning, landing system. A SWD has a base number indicating the logical system which it describes, and a dash number indicating its effectivity.

T**Termination Code**

Code indicating possible terminations at that point

U**Update**

Update wire process.

V**Vendor**

Approved supplier of materials and parts whose details appear in Vendor library.

W**W/L**

List of wires contained in a specific version of a bundle.

Wire Accommodation

The Wire Accommodation indicates the range of wires that a terminator can accommodate

Wire Length

The length of a wire connecting one part to another (including the route through a number of segments and branches).

Wire List (WL)

List of wires contained in a specific version of a bundle.

Wire Type

A 3-element code for describing type of wire; consists of family type, specification type and gauge, which are obtained from the bundle wire list data.

Work Package

Includes a group of instructions for maintenance and repair of the wire, or a list of tools needed for the task and a cross reference table for it

Work Package Set

Group of work packages for a system or sub-system

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